CFD EXPERTS

Simulate the Future

Ansys Fluent Workbench Tutorial Guide

Table of Contents

List of Figures

List of Tables

vii

Using This Manual

This preface is divided into the following sections:

1. What's In This Manual

The Ansys Fluent Workbench Tutorial Guide contains a number of tutorials that teach you how to use Ansys Fluent to solve different types of problems.

2. Typographical Conventions Used In This Manual

Several typographical conventions are used in this manual's text to help you find commands in the user interface.

- Different type styles are used to indicate graphical user interface items and text interface items. For example:

  Iso-Surface dialog box

  `surface/iso-surface` text command

- The text interface type style is also used when illustrating exactly what appears on the screen to distinguish it from the narrative text. In this context, user inputs are typically shown in boldface. For example,

```
solve/initialize/set-fmg-initialization

 Customize your FMG initialization:
   set the number of multigrid levels [5]

   set FMG parameters on levels ..

    residual reduction on level 1 is:  [0.001]
    number of cycles on level 1 is:  [10]  100

    residual reduction on level 2 is:  [0.001]
    number of cycles on level 2 is:  [50]  100
```

- Mini flow charts are used to guide you through the ribbon or the tree, leading you to a specific option, dialog box, or task page. The following tables list the meaning of each symbol in the mini flow charts.

Table 1: Mini Flow Chart Symbol Descriptions

Symbol	Indicated Action
	Look at the ribbon
	Look at the tree
	Double-click to open task page

≣	Select from task page
🖱→	Right-click the preceding item

For example,

▦ **Setting Up Domain** → **Mesh** → **Transform** → **Translate...**

indicates selecting the **Setting Up Domain** ribbon tab, clicking **Transform** (in the **Mesh** group box) and selecting **Translate...**, as indicated in the figure below:
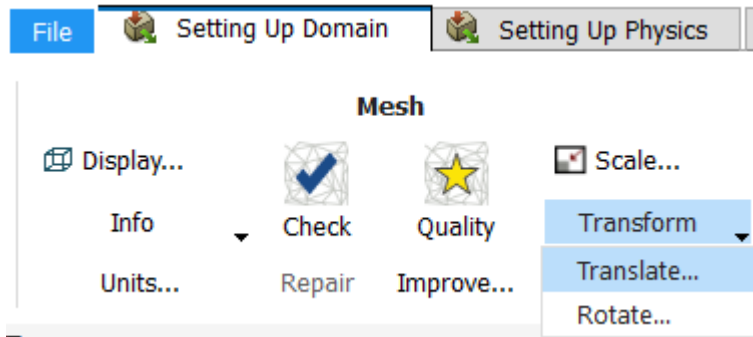
And

▦ **Setup** → **Models** → **Viscous** 🖱→ **Model** → **Realizable k-epsilon**

indicates expanding the **Setup** and **Models** branches, right-clicking **Viscous**, and selecting **Realizable k-epsilon** from the **Model** sub-menu, as shown in the following figure:
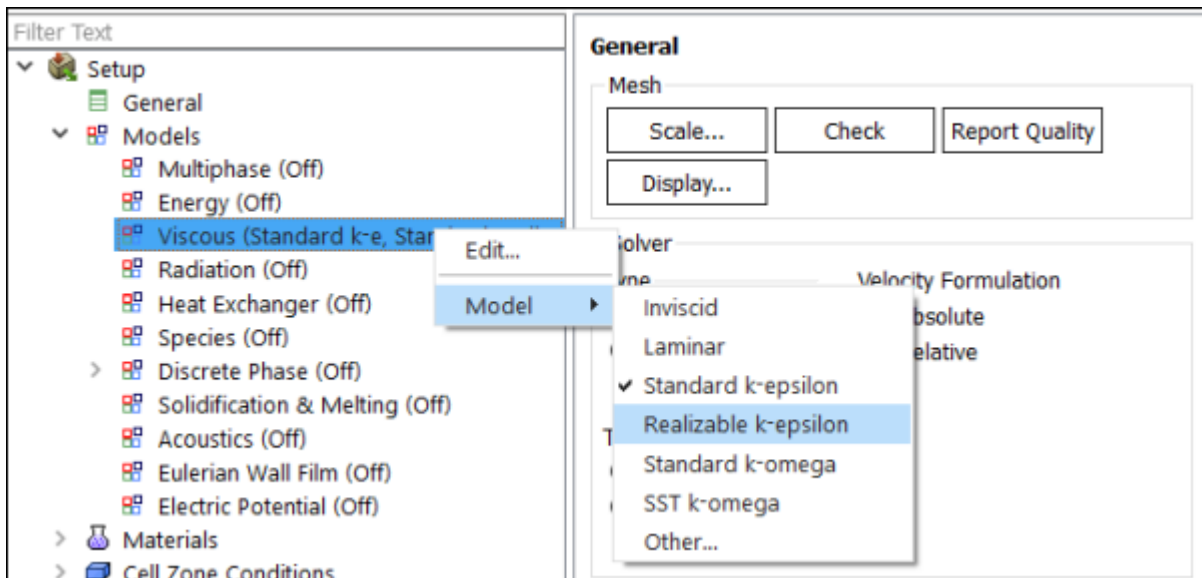
And

▦ **Setup** → ✛ **Boundary Conditions** → ≣ **velocity-inlet-5**

indicates opening the task page as shown below:
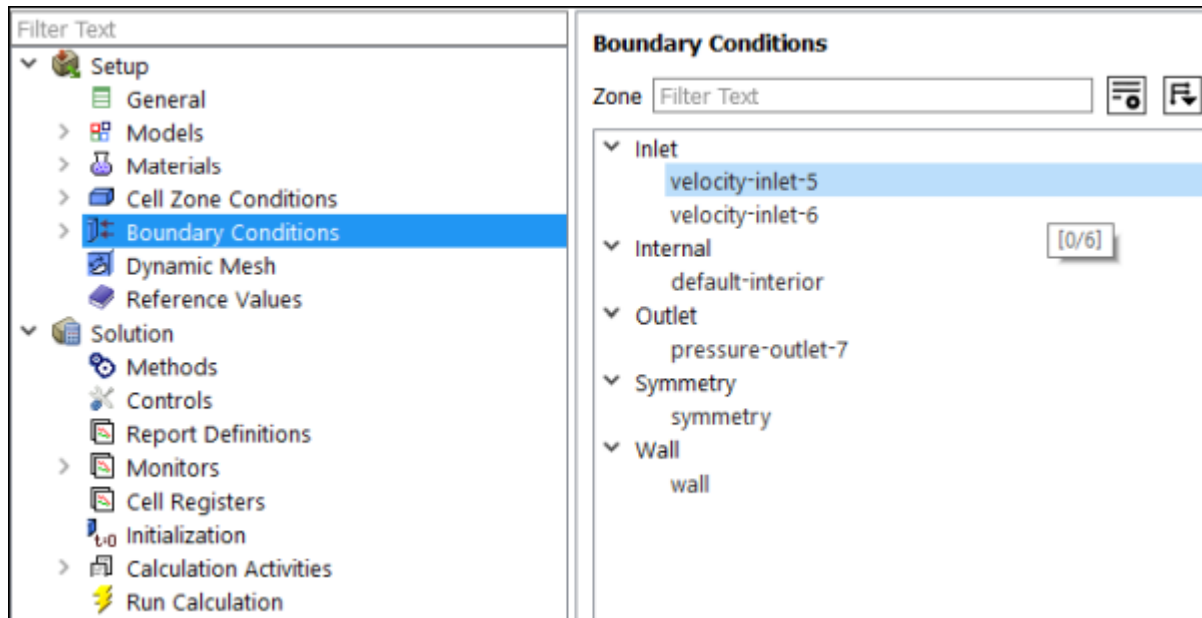
In this manual, mini flow charts usually accompany a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They show you how to quickly access a command or dialog box without having to search the surrounding material.

- In-text references to **File** ribbon tab selections can be indicated using a "/". For example **File/Write/Case...** indicates clicking the **File** ribbon tab and selecting **Case...** from the **Write** submenu (which opens the **Select File** dialog box).

Chapter 1: Introduction to Using Ansys Fluent in Ansys Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial is divided into the following sections:

1.1. Introduction

This tutorial illustrates using Ansys Fluent fluid flow systems in Ansys Workbench to set up and solve a three-dimensional turbulent fluid-flow and heat-transfer problem in a mixing elbow. It is designed to introduce you to the Ansys Workbench tool set using a simple geometry. Guided by the steps that follow, you will create the elbow geometry and the corresponding computational mesh using the geometry and meshing tools within Ansys Workbench. You will use Ansys Fluent to set up and solve the CFD problem, then visualize the results in both Ansys Fluent and in the CFD-Post postprocessing tool. Some capabilities of Ansys Workbench (for example, duplicating fluid flow systems, connecting systems, and comparing multiple data sets) are also examined in this tutorial.

This tutorial demonstrates how to do the following:

- Launch Ansys Workbench.

- Create a Fluent fluid flow analysis system in Ansys Workbench.

- Create the elbow geometry using Ansys DesignModeler.

- Create the computational mesh for the geometry using Ansys Meshing.

- Set up the CFD simulation in Ansys Fluent, which includes:

  - Setting material properties and boundary conditions for a turbulent forced-convection problem.

  - Initiating the calculation with residual plotting.

  - Calculating a solution using the pressure-based solver.

  - Examining the flow and temperature fields using Ansys Fluent and CFD-Post.

- Create a copy of the original Fluent fluid flow analysis system in Ansys Workbench.

- Change the geometry in Ansys DesignModeler, using the duplicated system.

- Regenerate the computational mesh.

- Recalculate a solution in Ansys Fluent.

- Compare the results of the two calculations in CFD-Post.

1.2. Prerequisites

This tutorial assumes that you have little to no experience with Ansys Workbench, Ansys DesignModeler, Ansys Meshing, Ansys Fluent, or CFD-Post, and so each step will be explicitly described.

1.3. Problem Description

The problem to be considered is shown schematically in Figure 1.1: Problem Specification (p. 3). A cold fluid at 293.15 K flows into the pipe through a large inlet and mixes with a warmer fluid at 313.15 K that enters through a smaller inlet located at the elbow. The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is often important to predict the flow field and temperature field in the area of the mixing region in order to properly design the junction.

> **Note:**
>
> Because the geometry of the mixing elbow is symmetric, only half of the elbow must be modeled.

Figure 1.1: Problem Specification
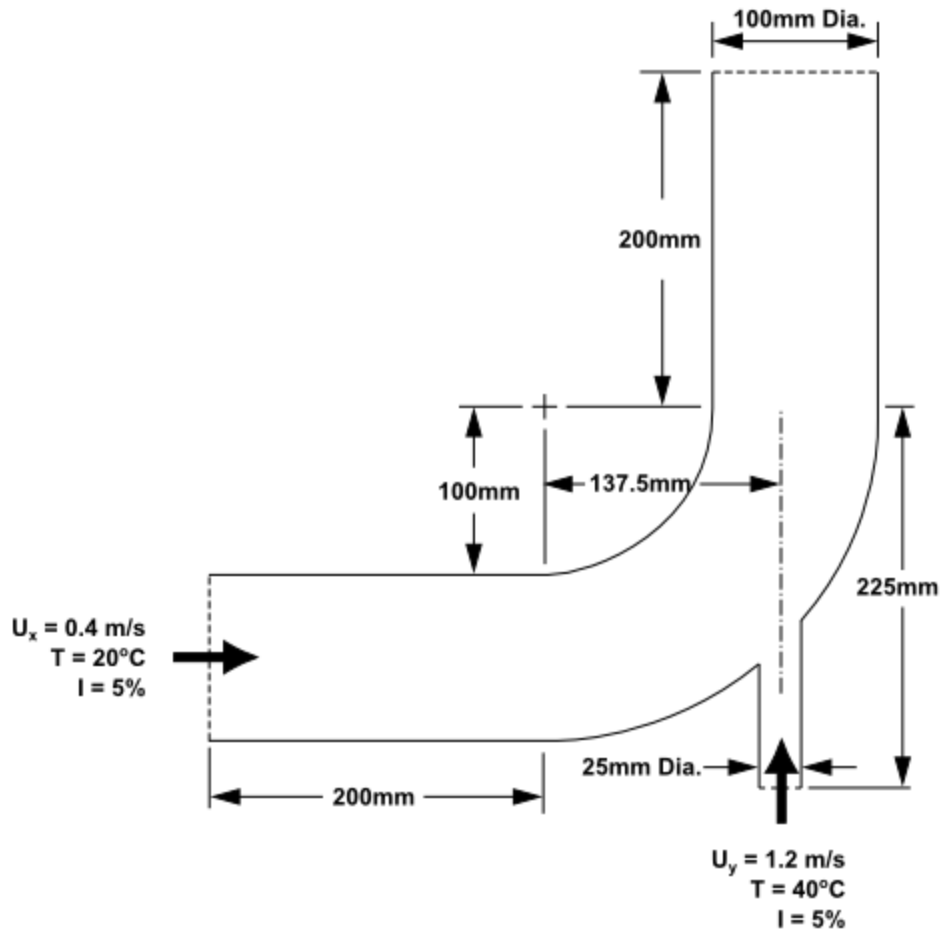
Note:

> The functionality to create named selections exists in both Ansys DesignModeler and Ansys Meshing. For the purposes of this tutorial, named selections are created in Ansys Meshing since the meshing application provides more comprehensive and extensive named selection functionality.

1.4. Setup and Solution

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the Ansys Fluent Tutorial Guide uses several type styles and mini flow charts. See Typographical Conventions Used In This Manual (p. ix) for detailed information.

The following sections describe the setup and solution steps for this tutorial:

1.4.1. Preparation

1. Download the `workbench_elbow.zip` file here.

2. Unzip `workbench_elbow.zip` to your working folder. This file contains a folder, `work-bench_elbow`, that holds the following items:

    - two geometry files, `elbow_geometry.agdb` and `elbow_geometry.stp`

    Note:

    Ansys Fluent tutorials are prepared using Ansys Fluent on a Windows system. The screen shots and graphic images in the tutorials may be slightly different than the appearance on your system, depending on the operating system or graphics card.

1.4.2. Creating a Fluent Fluid Flow Analysis System in Ansys Workbench

In this step, you will start Ansys Workbench, create a new Fluent fluid flow analysis system, then review the list of files generated by Ansys Workbench.

1. From the Windows **Start** menu, select **Start > ANSYS 2021 R2 > Workbench 2021 R2** to start a new Ansys Workbench session.

    This displays the Ansys Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox** and the components of the analysis system will be displayed in the **Project Schematic**.

    Note:

    Depending on which other products you have installed, the analysis systems that appear may differ from those in the figures that follow in this tutorial.

    ---

2. Create a new Fluent fluid flow analysis system by double-clicking the **Fluid Flow (Fluent)** option under **Analysis Systems** in the **Toolbox**.

> **Tip:**
>
> You can also drag-and-drop the analysis system into the **Project Schematic**. A green dotted outline indicating a potential location for the new system initially appears in the **Project Schematic**. When you drag the system to one of the outlines, it turns into a red box to indicate the chosen location of the new system.

Figure 1.2: Ansys Workbench with a New Fluent-Based Fluid Flow Analysis System
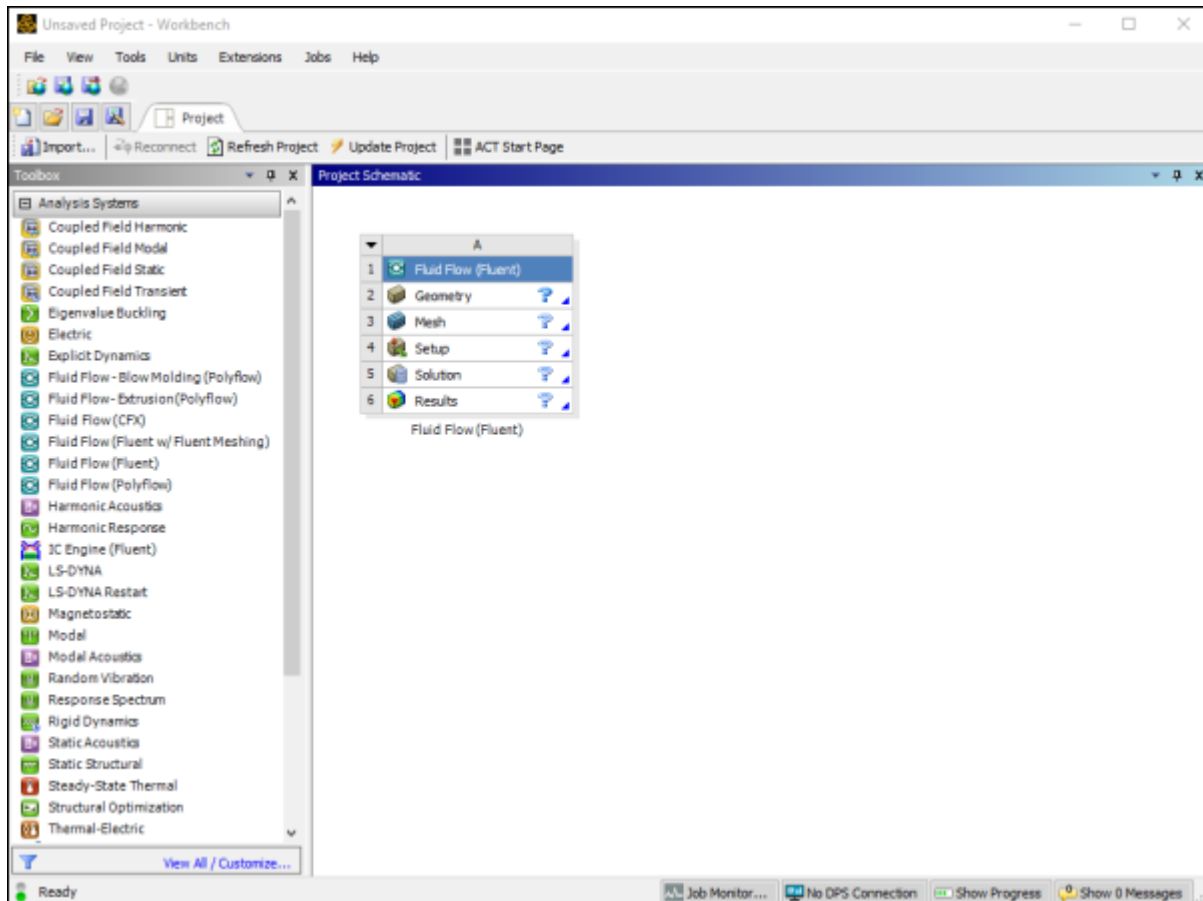
3. Name the analysis.

   a. Double-click the **Fluid Flow (Fluent)** label underneath the analysis system (if it is not already highlighted).

   b. Enter `elbow` for the name of the analysis system.

4. Save the project.

   a. In Ansys Workbench, under the **File** menu, select **Save**.

   File → Save

This displays the **Save As** dialog box, where you can browse to your working folder and enter a specific name for the Ansys Workbench project.

b. In your working directory, enter `elbow-workbench` as the project **File name** and click the **Save** button to save the project. Ansys Workbench saves the project with a `.wbpj` extension and also saves supporting files for the project.

Note that the fluid flow analysis system is composed of various *cells* (**Geometry**, **Mesh**, etc.) that represent the workflow for performing the analysis. Ansys Workbench is composed of multiple data-integrated and native applications in a single, seamless project flow, where individual cells can obtain data from other cells and provide data to other cells. As a result of this constant flow of data, a cell's state can quickly change. Ansys Workbench provides a visual indication of a cell's state at any given time via icons on the right side of each cell. Brief descriptions of the various states are provided below:

- **Unfulfilled** ( ) indicates that required upstream data does not exist. For example, when you first create a new **Fluid Flow (Fluent)** analysis system, all cells downstream of the **Geometry** cell appear as **Unfulfilled** because you have not yet specified a geometry for the system.

- **Refresh Required** ( ) indicates that upstream data has changed since the last refresh or update. For example, after you assign a geometry to the geometry cell in your new **Fluid Flow (Fluent)** analysis system, the **Mesh** cell appears as **Refresh Required** since the geometry data has not yet been passed from the **Geometry** cell to the **Mesh** cell.

- **Attention Required** ( ) indicates that the current upstream data has been passed to the cell, however, you must take some action to proceed. For example, after you launch Ansys Fluent from the **Setup** cell in a **Fluid Flow (Fluent)** analysis system that has a valid mesh, the **Setup** cell appears as **Attention Required** because additional data must be entered in Ansys Fluent before you can calculate a solution.

- **Update Required** ( ) indicates that local data has changed and the output of the cell must be regenerated. For example, after you launch Ansys Meshing from the **Mesh** cell in a **Fluid Flow (Fluent)** analysis system that has a valid geometry, the **Mesh** cell appears as **Update Required** because the **Mesh** cell has all the data it must generate an Ansys Fluent mesh file, but the Ansys Fluent mesh file has not yet been generated.

- **Up To Date** ( ) indicates that an update has been performed on the cell and no failures have occurred or that an interactive calculation has been completed successfully. For example, after Ansys Fluent finishes performing the number of iterations that you request, the **Solution** cell appears as **Up-to-Date**.

- **Interrupted, Update Required** ( ) indicates that you have interrupted an update (or canceled an interactive calculation that is in progress). For example, if you select the **Cancel** button in Ansys Fluent while it is iterating, Ansys Fluent completes the current iteration and then the **Solution** cell appears as **Interrupted, Update Required**.

- **Input Changes Pending** ( ) indicates that the cell is locally up-to-date, but may change when next updated as a result of changes made to upstream cells. For example, if you change the **Mesh** in an **Up-to-Date Fluid Flow (Fluent)** analysis system, the **Setup** cell

appears as **Refresh Required**, and the **Solution** and **Results** cells appear as **Input Changes Pending**.

- **Pending** () indicates that a batch or asynchronous solution is in progress. When a cell enters the **Pending** state, you can interact with the project to exit Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

For more information about cell states, see the Workbench User's Guide.

5. View the list of files generated by Ansys Workbench.

Ansys Workbench allows you to easily view the files associated with your project using the **Files** view. To open the **Files** view, select the **Files** option under the **View** menu at the top of the Ansys Workbench window.

View → Files

Figure 1.3: Ansys Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System

	A	B	C	D
1	Name ▼	Ce... ▼	Size ▼	Type ▼
2	🦀 elbow-workbench.wbpj		29 KB	Workbench Project File
3	☐ act.dat		259 KB	ACT Database
4	🔧 designPoint.wbdp		28 KB	Workbench Design Point File

In the **Files** view, you will be able to see the name and type of file, the ID of the cell that the file is associated with, the size of the file, the location of the file, and other information. For more information about the **Files** view, see the Workbench User's Guide.

> **Note:**
>
> The sizes of the files listed may differ slightly from those portrayed in Figure 1.3: Ansys Workbench Files View for the Project After Adding a Fluent-Based Fluid Flow Analysis System (p. 7).

From here, you will create the geometry described in Figure 1.1: Problem Specification (p. 3), and later create a mesh and set up a fluid flow analysis for the geometry.

1.4.3. Creating the Geometry in Ansys DesignModeler

For the geometry of your fluid flow analysis, you can create a geometry in Ansys DesignModeler, Ansys SpaceClaim Direct Modeler, or import the appropriate geometry file. In this step, you will create the geometry in Ansys DesignModeler, then review the list of files generated by Ansys Workbench.

Important:

Note the **Attention Required** icon ( ) within the **Geometry** cell for the system. This indicates that the cell requires data (for example, a geometry). Once the geometry is defined, the state of the cell will change accordingly. Likewise, the state of some of the remaining cells in the system will change.

Note:

If you would rather not create the geometry in Ansys DesignModeler, you can import a pre-existing geometry by right-clicking the **Geometry** cell and selecting the **Import Geometry** option from the context menu. From there, you can browse your file system to locate the `elbow_geometry.agdb` geometry file that is provided for this tutorial. If you do not have access to Ansys DesignModeler, you can use the `elbow_geometry.stp` file instead.

To learn how to create a mesh from the geometry you imported, go to Meshing the Geometry in the Ansys Meshing Application (p. 15).

---

1. Start Ansys DesignModeler.

   In the Ansys Workbench **Project Schematic**, right-click the **Geometry** cell in the `elbow` fluid flow analysis system to display the context menu, then select **New DesignModeler Geometry...**. This displays the Ansys DesignModeler application.

2. Set the units in Ansys DesignModeler.

   When Ansys DesignModeler first appears, you should select desired system of length units to work from. For the purposes of this tutorial (where you will create the geometry in millimeters and perform the CFD analysis using SI units) set the units to **Millimeter**.

   Units → Millimeter

3. Create the geometry.

   The geometry for this tutorial (Figure 1.1: Problem Specification (p. 3)) consists of a large curved pipe accompanied by a smaller side pipe. Ansys DesignModeler provides various geometry primitives that can be combined to rapidly create geometries such as this one. You will perform the following tasks to create the geometry:

   • Create the bend in the main pipe by defining a segment of a torus.

   • Extrude the faces of the torus segment to form the straight inlet and outlet lengths.

   • Create the side pipe by adding a cylinder primitive.

- Use the symmetry tool to reduce the model to half of the pipe assembly, thus reducing computational cost.

a. Create the main pipe:

   i. Create a new torus for the pipe bend by choosing the **Create → Primitives → Torus** menu item from the menubar.

   A preview of the torus geometry will appear in the graphics window. Note that this is a preview and the geometry has not been created yet. First you must specify the parameters of the torus primitive in the next step.

   ii. In the **Details View** for the new torus (**Torus1**), set **Base Y Component** to −1 by clicking the **1** to the right of **FD10, Base Y Component**, entering −1, and pressing **Enter**. This specifies the direction vector from the origin to the center of the circular cross-section at the start of the torus. In the same manner, specify **Angle**; **Inner Radius**; and **Outer Radius** as shown below.
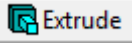
> **Note:**
>
> Enter only the value without the units of mm. They will be appended automatically because you specified the units previously.

Details of Torus1	
Torus	Torus1
Base Plane	XYPlane
Operation	Add Material
Origin Definition	Coordinates
☐ FD3, Origin X Coordinate	0 mm
☐ FD4, Origin Y Coordinate	0 mm
☐ FD5, Origin Z Coordinate	0 mm
Axis Definition	Components
☐ FD6, Axis X Component	0
☐ FD7, Axis Y Component	0
☐ FD8, Axis Z Component	1
Base Definition	Components
☐ FD9, Base X Component	0
☐ FD10, Base Y Component	-1
☐ FD11, Base Z Component	0
☐ FD12, Angle (>0)	90 °
☐ FD13, Inner Radius (>0)	100 mm
☐ FD14, Outer Radius (>0)	200 mm
As Thin/Surface?	No

   iii. To create the torus segment, click the **Generate** button ⚡Generate that is located in the Ansys DesignModeler toolbar.
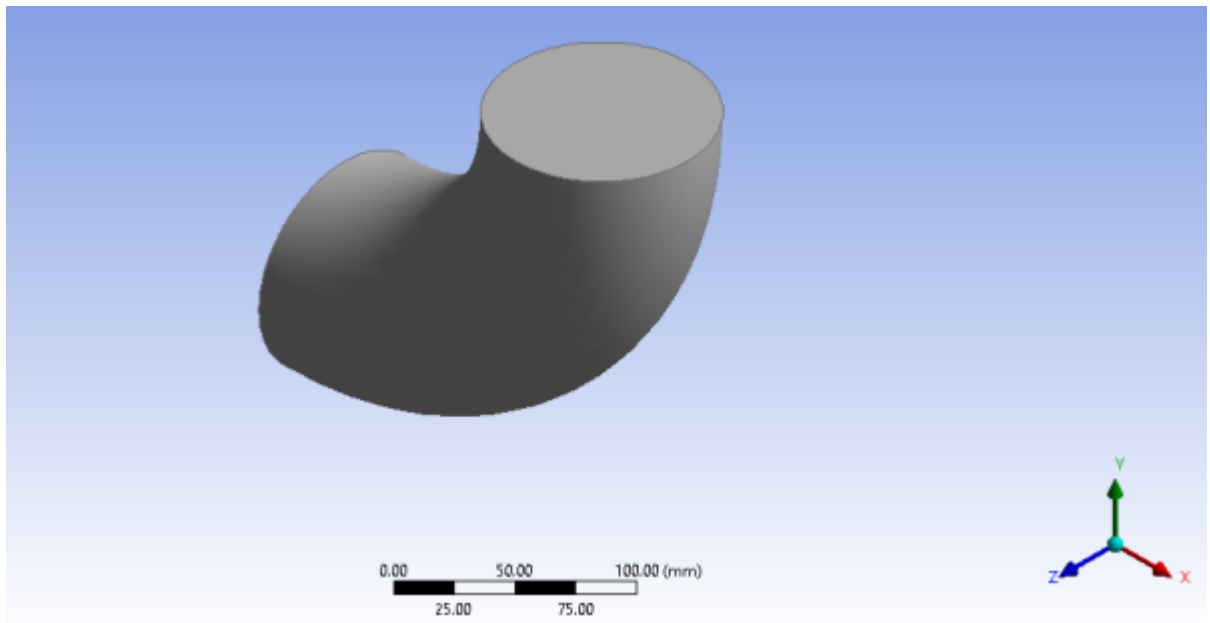
The `Torus1` item appears in the **Tree Outline** view. If you want to delete this item, you can right-click it and select **Delete** from the context menu that opens.

iv. Ensure that the selection filter is set to **Faces**. This is indicated by the **Faces** button
appearing depressed in the toolbar and the appearance of the Face selection cursor,
when you mouse over the geometry.

v. Select the top face (in the positive Y direction) of the elbow and click the **Extrude** button
from the **3D Features** toolbar.

vi. In the **Details View** for the new extrusion (**Extrude1**), click **Apply** to the right of **Geometry**. This accepts the face you selected as the base geometry of the extrusion.

vii. Click **None (Normal)** to the right of **Direction Vector**. Again, ensure that the selection filter is set to **Faces**, select the same face on the elbow to specify that the extrusion will be normal to the face and click **Apply**.
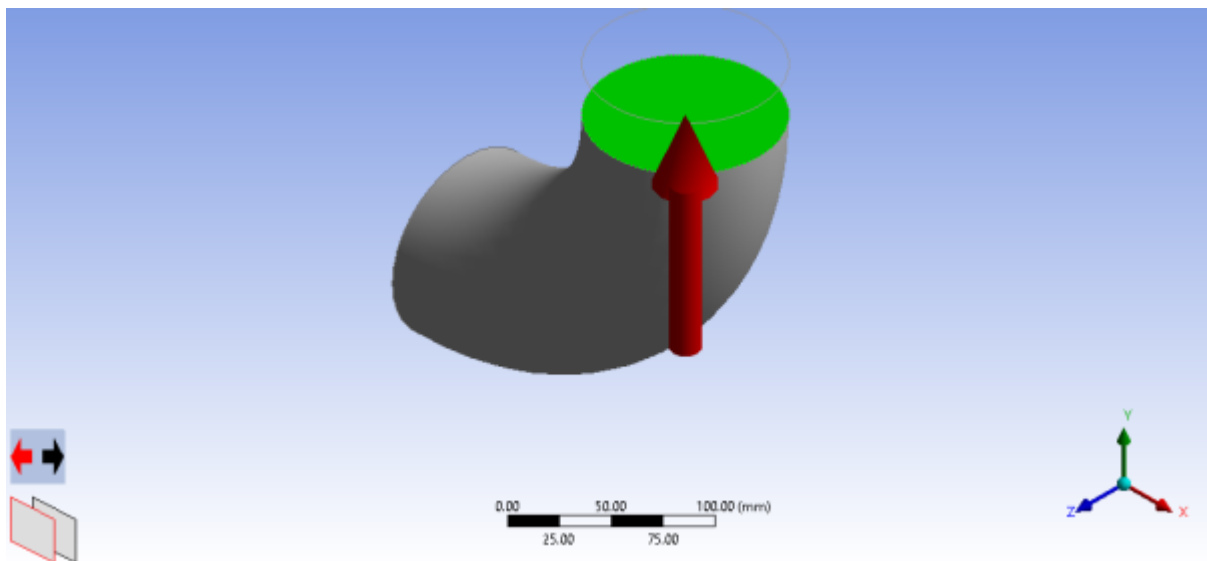
viii. Enter 200 for **FD1, Depth (>0)** and click **Generate**.
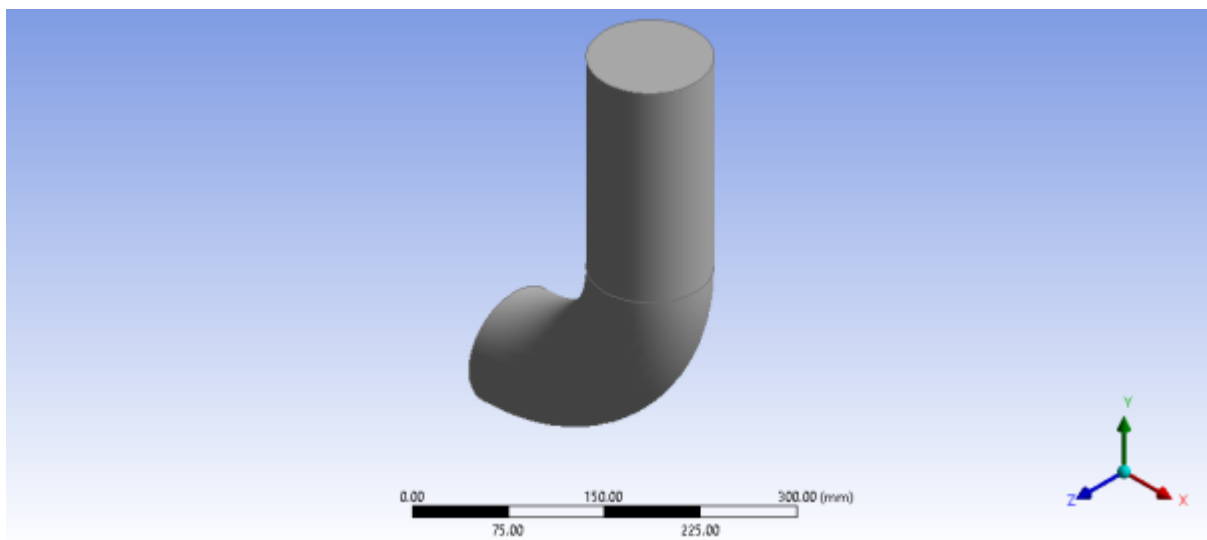
ix. In a similar manner, create an extrusion of the other face of the torus segment to create the 200 mm inlet extension. You will probably find it helpful to rotate the view so that you can easily select the other face of the bend.

You can use the mouse buttons to change your view of the 3D image. The following table describes mouse actions that are available:

Table 1.1: DesignModeler View Manipulation Instructions

Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, horizontal)	After clicking the **Rotate** icon, ⟳ , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.

Action	Using Graphics Toolbar Buttons and the Mouse
Translate or pan view	After clicking the **Pan** icon, ✛ , press and hold the left mouse button and drag the object with the mouse until the view is satisfactory.
Zoom in and out of view	After clicking the **Zoom** icon, ⊕ , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.
Box zoom	After clicking the **Box Zoom** icon, ⊕ , press and hold the left mouse button and drag the mouse diagonally across the screen. This action will cause a rectangle to appear in the display. When you release the mouse button, a new view will be displayed that consists entirely of the contents of the rectangle.

Clicking the **Zoom to Fit** icon, 🔍 , will cause the object to fit exactly and be centered in the window.

After entering the extrusion parameters and clicking **Generate**, the geometry should appear as in Figure 1.4: Elbow Main Pipe Geometry (p. 12).

Figure 1.4: Elbow Main Pipe Geometry
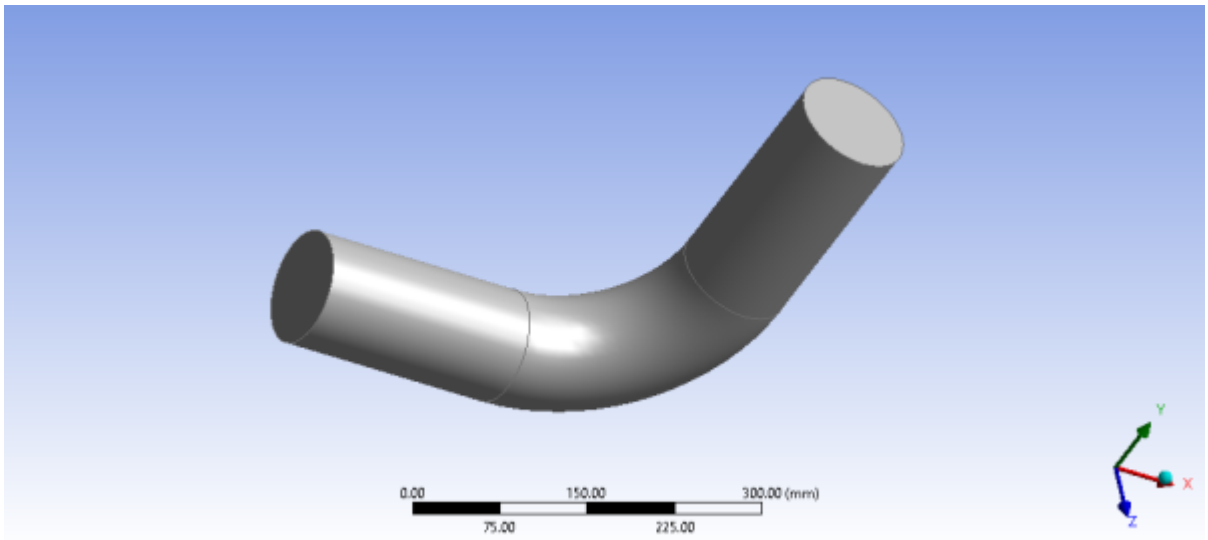
b. Next you will use a cylinder primitive to create the side pipe.

   i. Choose **Create** → **Primitives** → **Cylinder** from the menubar.

   ii. In the **Details View**, set the parameters for the cylinder as follows and click **Generate**:

Setting	Value
BasePlane	XYPlane
FD3, Origin X Coordinate	137.5
FD4, Origin Y Coordinate	-225

Setting	Value
FD5, Origin Z Coordinate	0
FD6, Axis X Component	0
FD7, Axis Y Component	125
FD8, Axis Z Component	0
FD10, Radius (>0)	12.5

The Origin Coordinates determine the starting point for the cylinder and the Axis Components determine the length and orientation of the cylinder body.
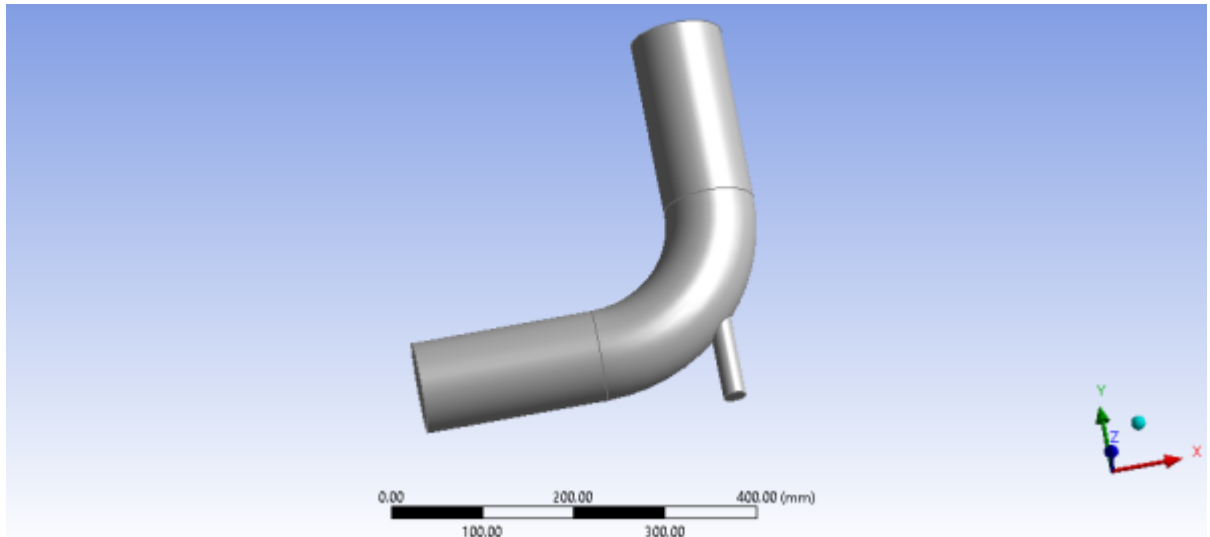
c. The final step in creating the geometry is to split the body on its symmetry plane which will halve the computational domain.

    i. Choose **Tools** → **Symmetry** from the menu bar.

    ii. Select the **XYPlane** in the **Tree Outline**.

    iii. Click **Apply** next to **Symmetry Plane 1** in the **Details** view.

    iv. Click **Generate**.
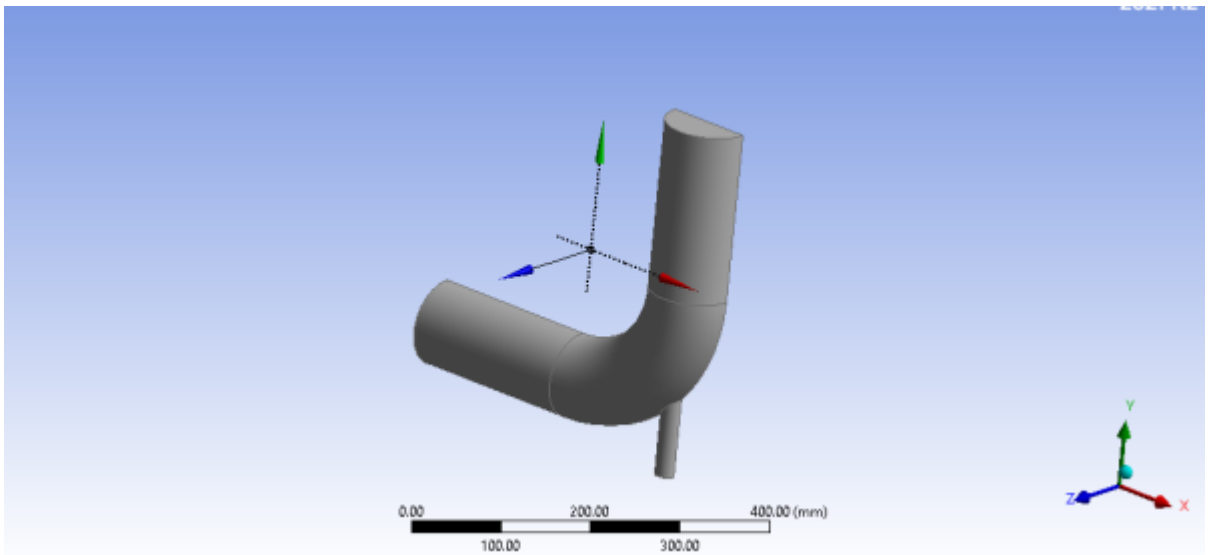
The new surface created with this operation will be assigned a symmetry boundary condition in Fluent so that the model will accurately reflect the physics of the complete elbow geometry even though only half of it is meshed.

d. Specify the geometry as a fluid body.

   i. In the **Tree Outline**, open the **1 Part, 1 Body** branch and select **Solid**.

   ii. In the **Details View** of the body, change the name of the **Body** from **Solid** to **Fluid**.

   iii. In the **Fluid/Solid** section, select **Fluid** from the drop-down list.

Details View	⊓
Details of Body	
Body	Fluid
Volume	2.5159e+006 mm³
Surface Area	1.7636e+005 mm²
Faces	8
Edges	18
Vertices	12
Fluid/Solid	Fluid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

   iv. Click **Generate**.

Tip:

In addition to the primitives you used in this tutorial, Ansys DesignModeler offers a full suite of 2D sketching and 3D solid modeling tools for creating arbitrary geometry. Refer to the DesignModeler User's Guide for more information.

4. Close Ansys DesignModeler by selecting **File** → **Close DesignModeler** or by clicking the 'X' icon in the upper right-hand corner. Ansys Workbench automatically saves the geometry and updates the **Project Schematic** accordingly. The question mark in the **Geometry** cell is replaced by a check mark, indicating that there is a geometry now associated with the fluid flow analysis system.

5. View the list of files generated by Ansys Workbench by selecting **View** → **Files**.

Figure 1.5: Ansys Workbench Files View for the Project After Creating the Geometry

Note the addition of the geometry file (`FFF.agdb`, where `FFF` indicates a Fluent-based fluid flow system) to the list of files. If you had imported the geometry file provided for this tutorial rather than creating the geometry yourself, the `elbow_geometry.agdb` (or the `elbow_geometry.stp`) file would be listed instead.

1.4.4. Meshing the Geometry in the Ansys Meshing Application

Now that you have created the mixing elbow geometry, you must generate a computational mesh throughout the flow volume. For this section of the tutorial, you will use the Ansys Meshing application to create a mesh for your CFD analysis, then review the list of files generated by Ansys Workbench.

Important:

Note the **Refresh Required** icon (↻) within the **Mesh** cell for the system. This indicates that the state of the cell requires a refresh and that upstream data has changed since the last refresh or update (such as an update to the geometry). Once the mesh is defined, the state of the **Mesh** cell will change accordingly, as will the state of the next cell in the system, in this case the **Setup** cell.
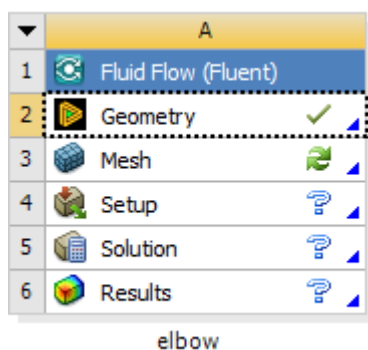
1. Open the Ansys Meshing application.

In the Ansys Workbench **Project Schematic**, double-click the **Mesh** cell in the `elbow` fluid flow analysis system (cell A3). This displays the Ansys Meshing application with the elbow geometry already loaded. You can also right-click the **Mesh** cell to display the context menu where you can select the **Edit...** option.

Figure 1.6: The Ansys Meshing Application with the Elbow Geometry Loaded
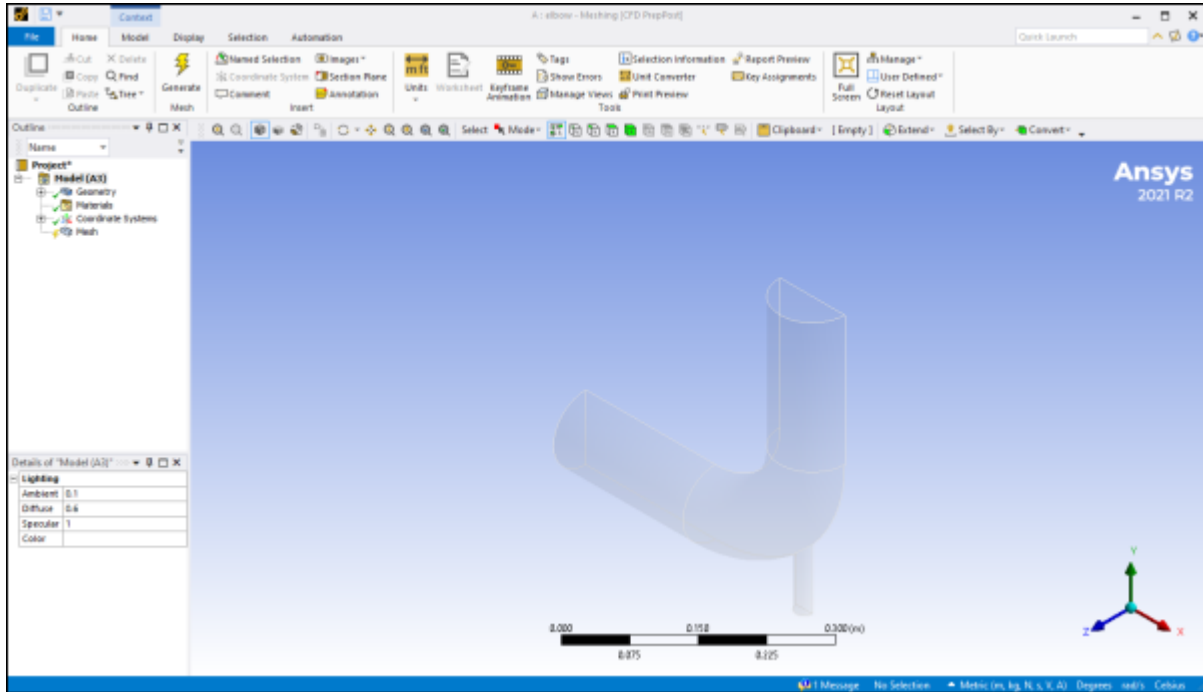
2. Create named selections for the geometry boundaries.

   In order to simplify your work later on in Ansys Fluent, you should label each boundary in the geometry by creating named selections for the pipe inlets, the outlet, and the symmetry surface (the outer wall boundaries are automatically detected by Ansys Fluent).

   a. Select the large inlet in the geometry that is displayed in the Ansys Meshing application.

   > **Tip:**
   >
   > • Use the Graphics Toolbar buttons and the mouse to manipulate the image until you can easily see the pipe openings and surfaces.
   >
   > • To select the inlet, the *Single select* ( ) mode must be active.

   b. Right-click and select the **Create Named Selection** option.

Figure 1.7: Selecting a Face to Name
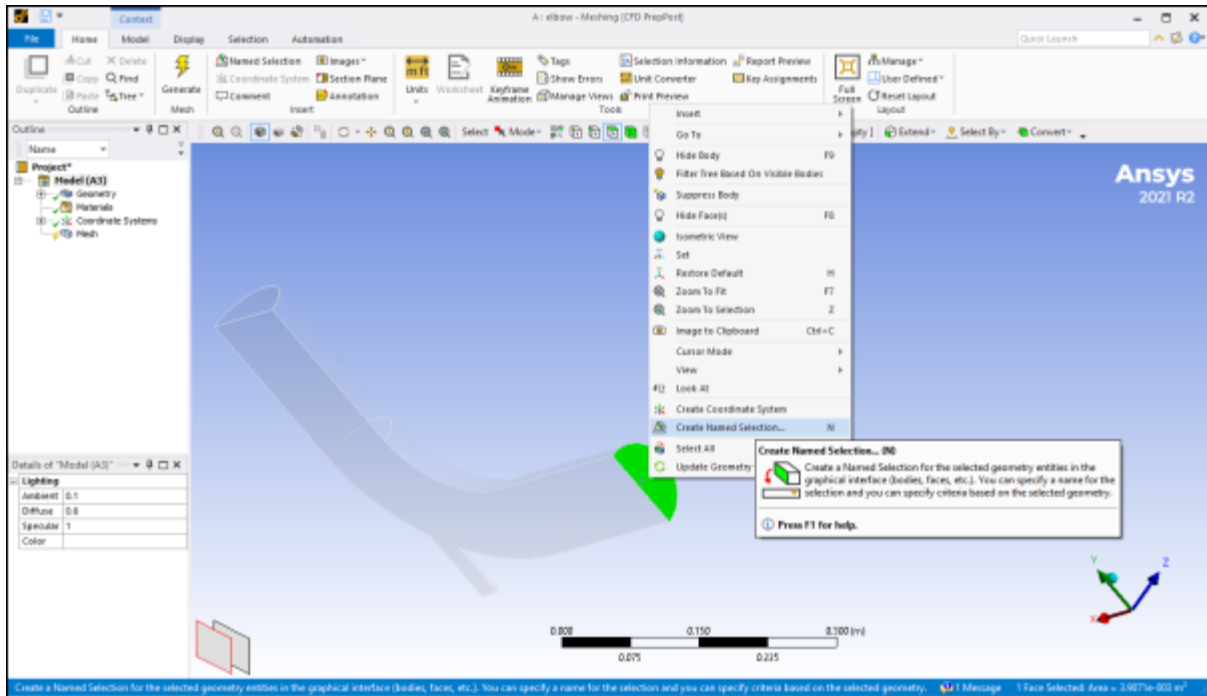
This displays the **Selection Name** dialog box.

Figure 1.8: Applying a Name to a Selected Face
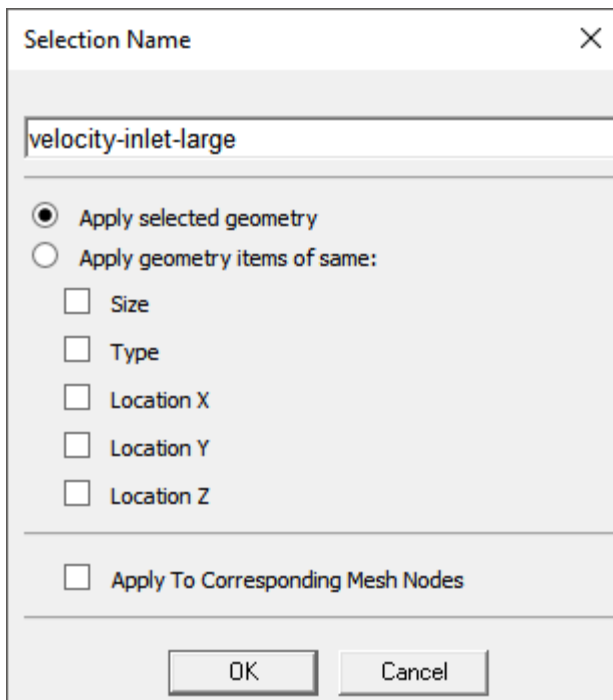
c. In the **Selection Name** dialog box, enter `velocity-inlet-large` for the name and click **OK**.

d. Perform the same operations for:

- The small inlet (`velocity-inlet-small`)

- The large outlet (`pressure-outlet`)

- The symmetry plane (`symmetry`).

The named selections that you have created appear under the **Named Selections** item in the **Outline** view.

Important:

It is important to note that by using the strings "velocity inlet" and "pressure outlet" in the named selections (with or without hyphens or underscore characters), Ansys Fluent automatically detects and assigns the corresponding boundary types accordingly.

---

3. Create a named selection for the fluid body.

   a. Change the selection filter to **Body** in the **Graphics Toolbar** ( ).

   b. Click the elbow in the graphics display to select it.

   c. Right-click, select the **Create Named Selection** option and name the body `Fluid`.

   By creating a named selection called `Fluid` for the fluid body you will ensure that Ansys Fluent automatically detects that the volume is a fluid zone and treats it accordingly.

4. Set basic meshing parameters for the Ansys Meshing application.

   For this analysis, you will adjust several meshing parameters to obtain a finer mesh.

   a. In the **Outline** view, select **Mesh** under **Project/Model** to display the **Details of "Mesh"** view below the **Outline** view.

Important:

Note that because the Ansys Meshing application automatically detects that you are going to perform a CFD fluid flow analysis using Ansys Fluent, the **Physics Preference** is already set to **CFD** and the **Solver Preference** is already set to **Fluent**.

---

   b. Expand the **Quality** node to reveal additional quality parameters. Change **Smoothing** to **High**.

   c. Add a **Sizing** control.

      i. With **Mesh** still selected in the **Outline** tree, click the elbow in the graphics display to select it.

      ii. Right click in the graphics area and select **Insert** → **Sizing** from the context menu.
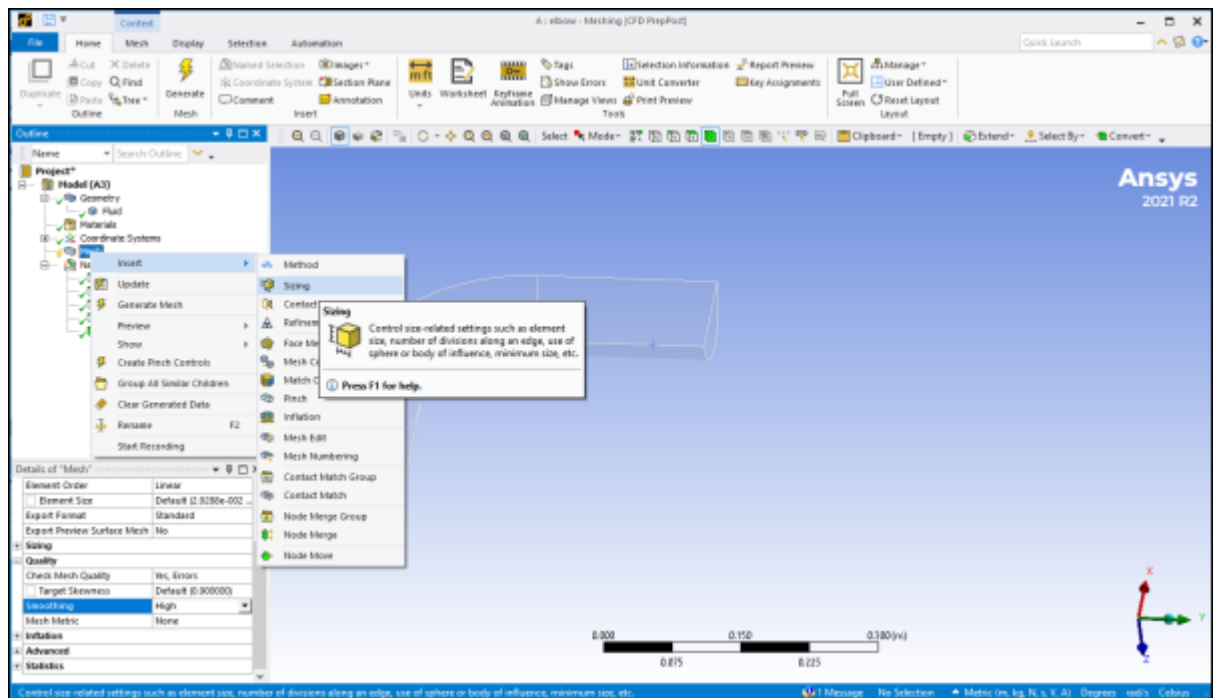
*A new **Sizing** entry appears under **Mesh** in the project **Outline** tree*

    iii. In the **Outline** tree, click the new **Sizing** control.

    iv. Enter `6e-3` for **Element Size** and press **Enter**.

  d. Click again **Mesh** in the **Outline** view and in the **Details of "Mesh"** view, expand the **Inflation** node to reveal additional inflation parameters. Change **Use Automatic Inflation** to **Program Controlled**.

5. Generate the mesh.

    Right-click **Mesh** in the project **Outline** tree, and select **Update** in the context menu.

Figure 1.9: The Computational Mesh for the Elbow Geometry in the Ansys Meshing Application
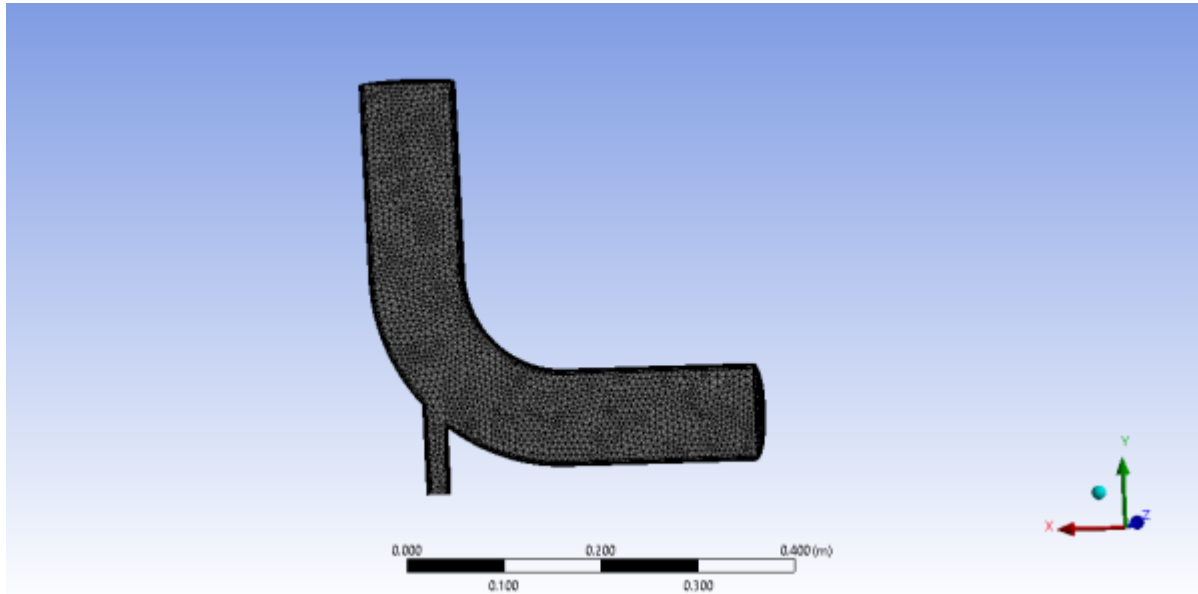
Important:

Using the **Generate Mesh** option creates the mesh, but does not actually create
the relevant mesh files for the project and is optional if you already know that the
mesh is acceptable. Using the **Update** option automatically generates the mesh,
creates the relevant mesh files for your project, and updates the Ansys Workbench
cell that references this mesh.

Note:

Once the mesh is generated, you can view the mesh statistics by opening the **Statistics**
node in the **Details of "Mesh"** view. This will display information such as the number
of nodes and the number of elements.

6. Close the Ansys Meshing application.

   You can close the Ansys Meshing application without saving it because Ansys Workbench auto-
   matically saves the mesh and updates the **Project Schematic** accordingly. The **Refresh Required**
   icon in the **Mesh** cell has been replaced by a check mark, indicating that there is a mesh now
   associated with the fluid flow analysis system.

7. View the list of files generated by Ansys Workbench.

   View → Files

Figure 1.10: Ansys Workbench Files View for the Project After Mesh Creation

	A	B	C	D
1	Name ▼	Ce... ▼	Size ▼	Type ▼
2	FFF.agdb	A2	2 MB	Geometry File
3	elbow-workbench.wbpj		35 KB	Workbench Project File
4	act.dat		259 KB	ACT Database
5	FFF.mshdb	A3	6 MB	Mesh Database File
6	designPoint.wbdp		31 KB	Workbench Design Point File
7	FFF.msh	A3	455 KB	Fluent Mesh File

Note the addition of the mesh files (`FFF.msh` and `FFF.mshdb`) to the list of files. The `FFF.msh` file is created when you update the mesh, and the `FFF.mshdb` file is generated when you close the Ansys Meshing application.

1.4.5. Setting Up the CFD Simulation in Ansys Fluent

Now that you have created a computational mesh for the elbow geometry, you will set up a CFD analysis using Ansys Fluent, then review the list of files generated by Ansys Workbench.

1.4.5.1. Starting Ansys Fluent

1. In the Ansys Workbench **Project Schematic**, double-click the **Setup** cell in the `elbow` fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

   When Ansys Fluent is first started, the Fluent Launcher is displayed, enabling you to view and/or set certain Ansys Fluent start-up options.

   > **Note:**
   >
   > The Fluent Launcher allows you to decide which version of Ansys Fluent you will use, based on your geometry and on your processing capabilities.

Figure 1.11: Fluent Launcher
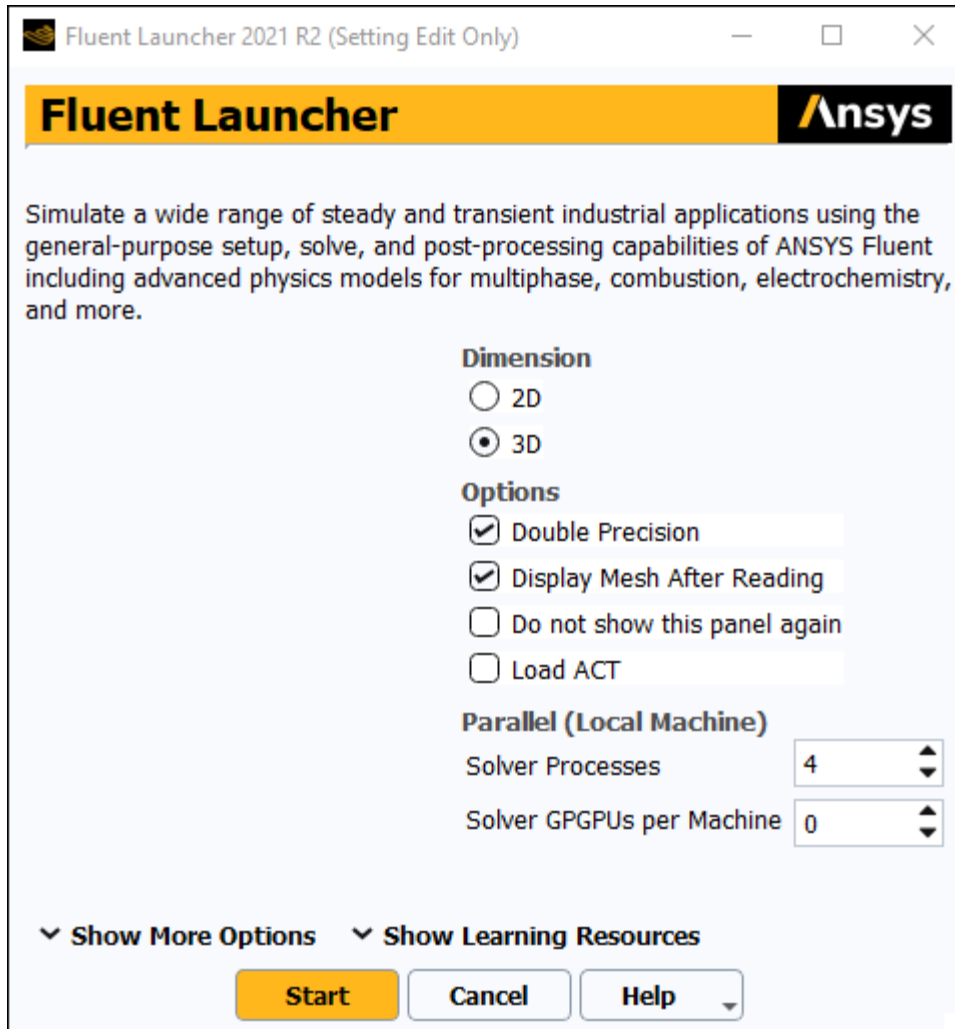
2. Ensure that the proper options are enabled.

Important:

Note that the **Dimension** setting is already filled in and cannot be changed, since Ansys Fluent automatically sets it based on the mesh or geometry for the current system.

---

a. Ensure that the **Display Mesh After Reading** option is enabled.

b. Select **Double Precision** under **Options**.

c. Set **Solver Processes** to 4 under **Parallel (Local Machine)**.

Note:

Fluent will retain your preferences for future sessions.

---

3. Click **Start** to launch Ansys Fluent.

> **Note:**
>
> The Ansys Fluent settings file (`FFF.set`) is written as soon as Ansys Fluent opens.

Figure 1.12: The Ansys Fluent Application
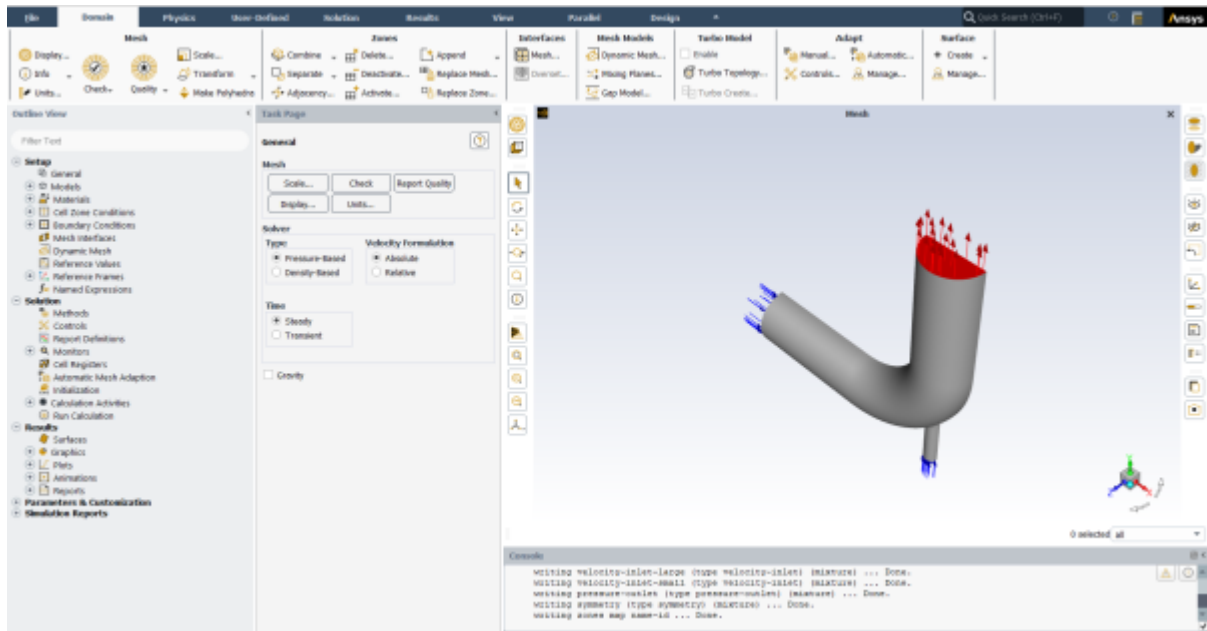
1.4.5.2. Setting Up Domain

In this step, you will perform the mesh-related activities using the **Domain** ribbon tab (**Mesh** group).
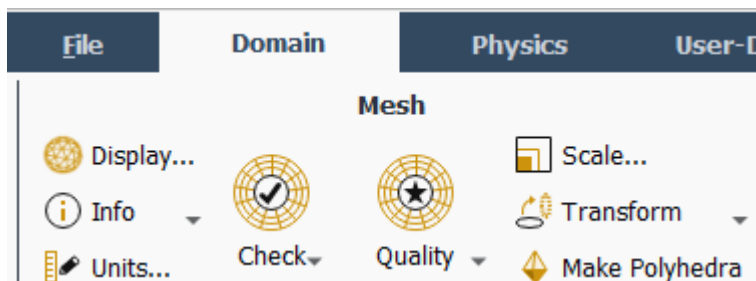
> **Note:**
>
> These controls are also available in the **General** task page that can be accessed by clicking the **Setup/General** tree item.

1. Change the units for length.

Because you want to specify and view values based on a unit of length in millimeters from within Ansys Fluent, change the units of length within Ansys Fluent from meters (the default) to millimeters.

Important:

Note that the Ansys Meshing application automatically converts and exports meshes for Ansys Fluent using meters (m) as the unit of length regardless of what units were used to create them. This is so you do not have to scale the mesh in Ansys Fluent under Ansys Workbench.

Domain → **Mesh** → **Units...**

This displays the **Set Units** dialog box.
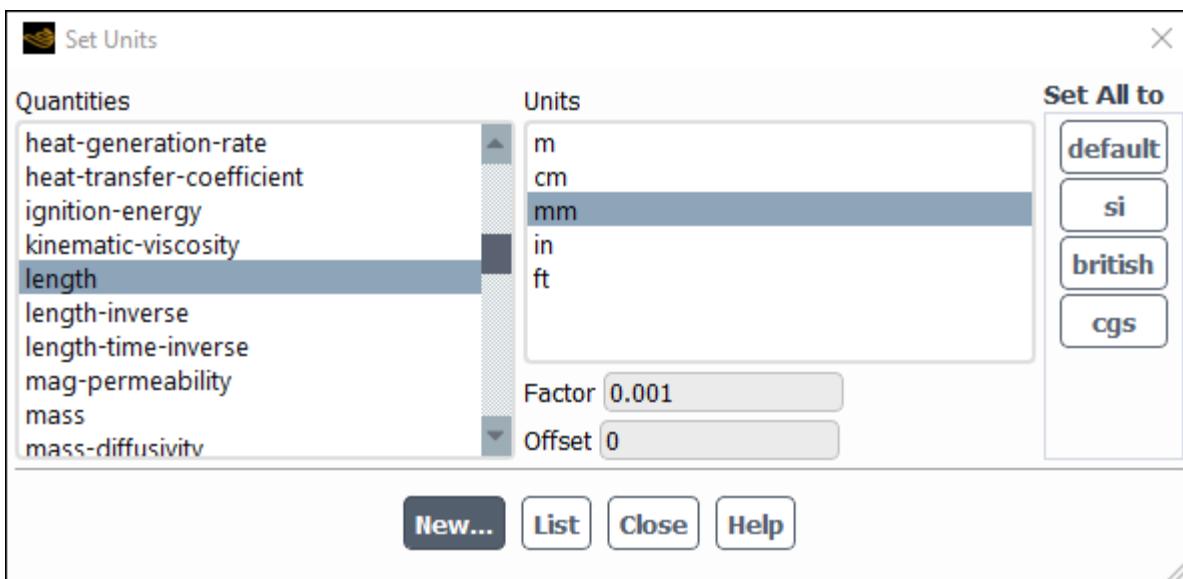
a. Select **length** in the **Quantities** list.

b. Select **mm** in the **Units** list.

c. Close the dialog box.

Note:

Now, all subsequent inputs that require a value based on a unit of length can be specified in millimeters rather than meters.

---

2. Check the mesh.

Domain → Mesh → Check → Perform Mesh Check

> **Note:**
>
> Ansys Fluent will report the results of the mesh check in the console.

```
 Domain Extents:
   x-coordinate: min (m) = -2.000000e-01, max (m) = 2.000000e-01
   y-coordinate: min (m) = -2.250000e-01, max (m) = 2.000000e-01
   z-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e-02
 Volume statistics:
   minimum volume (m3): 2.836821e-09
   maximum volume (m3): 6.611039e-08
     total volume (m3): 2.510838e-03
 Face area statistics:
   minimum face area (m2): 1.854435e-06
   maximum face area (m2): 4.386630e-05
 Checking mesh....................................
Done.
```

> **Note:**
>
> The minimum and maximum values may vary slightly when running on different
> platforms. The mesh check will list the minimum and maximum x and y values from
> the mesh in the default SI unit of meters. It will also report a number of other mesh
> features that are checked. Any errors in the mesh will be reported at this time. Ensure
> that the minimum volume is not negative as Ansys Fluent cannot begin a calculation
> when this is the case.

3. Review the mesh quality.

Domain → Mesh → Quality → Evaluate Mesh Quality

> **Note:**
>
> Ansys Fluent will report the results of the mesh quality below the results of the mesh
> check in the console.

```
Mesh Quality:

Minimum Orthogonal Quality =  2.03636e-01 cell 19646 on zone 4 (ID: 115652 on partition: 1)
at location ( 1.68524e-01,  1.01318e-01,  5.41369e-04)

Maximum Aspect Ratio =  3.09204e+01 cell 1395 on zone 4 (ID: 508 on partition: 0)
at location ( 1.25755e-01, -1.56954e-01,  3.15732e-03)
```

> **Note:**
>
> The quality of the mesh plays a significant role in the accuracy and stability of the
> numerical computation. Checking the quality of your mesh is, therefore, an important
> step in performing a robust simulation. Minimum cell orthogonality is an important
> indicator of mesh quality. Values for orthogonality can vary between 0 and 1 with

> lower values indicating poorer quality cells. In general, the minimum orthogonality
> should not be below 0.01 with the average value significantly larger. The high aspect
> ratio cells in this mesh are near the walls and are a result of the boundary layer infla-
> tion applied in the meshing step. For more information about the importance of
> mesh quality refer to the Fluent User's Guide.

1.4.5.3. Setting Up Physics

In the steps that follow, you will select a solver and specify physical models, material properties, and zone conditions for your simulation using the **Setting Up Physics** ribbon tab.

1. In the **Solver** group of the **Physics** ribbon tab, retain the default selection of the steady pressure-based solver.

   Physics → Solver
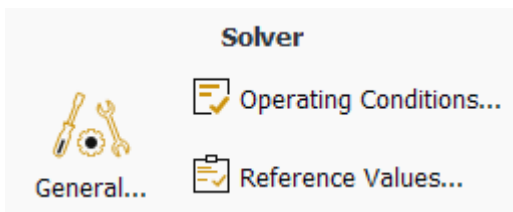
2. Set up your models for the CFD simulation using the **Models** group of the **Physics** ribbon tab.
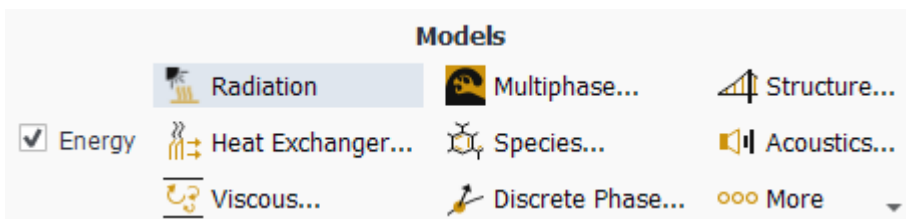
   > **Note:**
   >
   > You can also use the **Models** task page that can be accessed by double-clicking the **Setup/Models** tree item.

   a. Enable heat transfer by activating the energy equation.

      In the **Physics** ribbon tab, select **Energy** (**Models** group).

      Physics → Models → Energy

   b. Enable the k-ω SST turbulence model.

      i. In the **Physics** ribbon tab, click **Viscous...** (**Models** group).

         Physics → Models → Viscous...
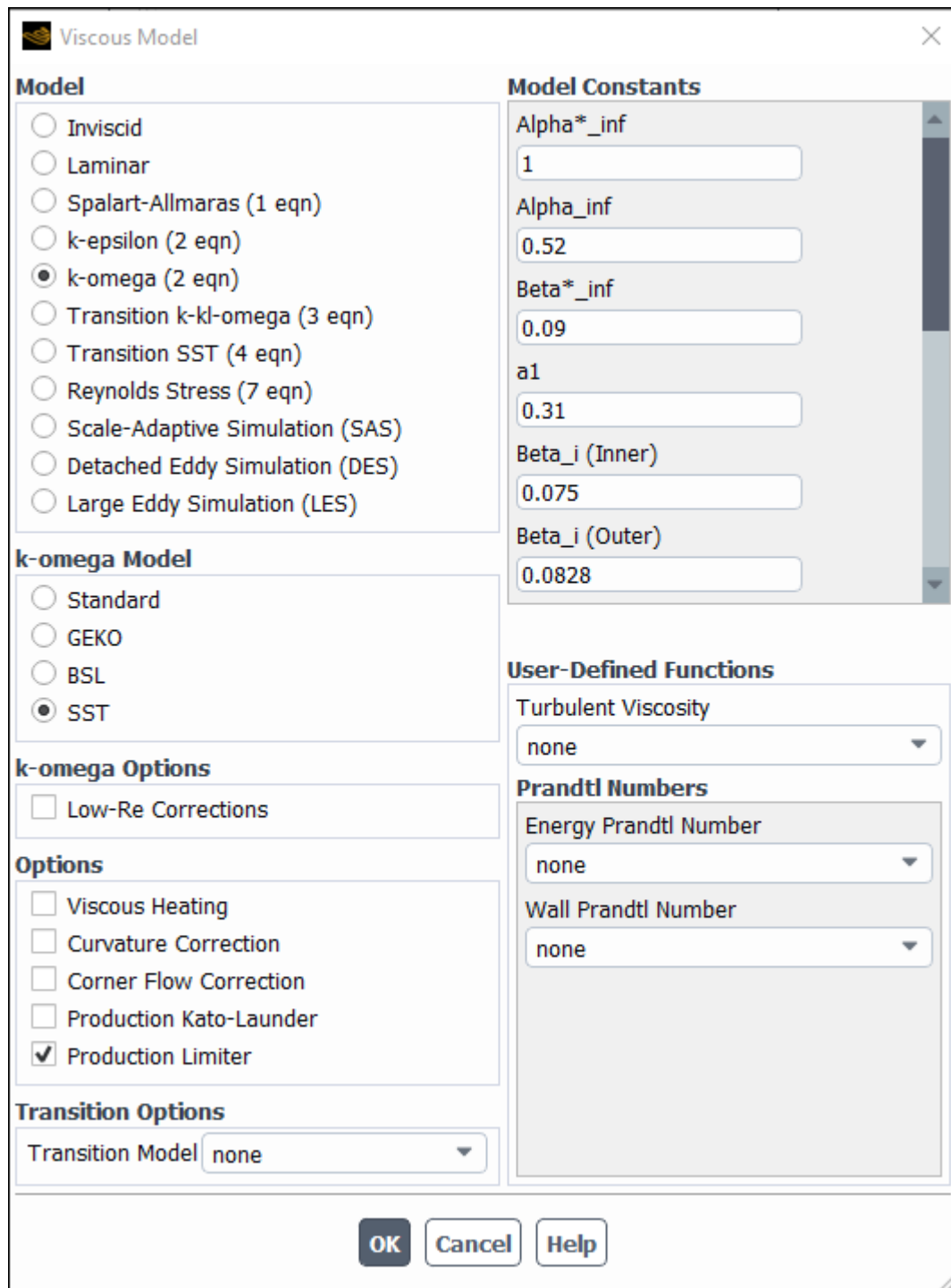
ii. In the **Viscous Model** dialog box, retain the default selection of **k-omega** from the **Model** list.

iii. Use the default **SST** from the **k-omega Model** group.

iv. Click **OK** to accept the model and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled.

3. Set up the materials for your CFD simulation using the **Materials** group of the **Physics** ribbon tab.

Materials

Create/Edit...

Create a new material called **water-liquid** using the **Create/Edit Materials** dialog box.

a. In the **Setting Up Physics** ribbon tab, click **Create/Edit...** (**Materials** group).

Physics → Materials → Create/Edit...
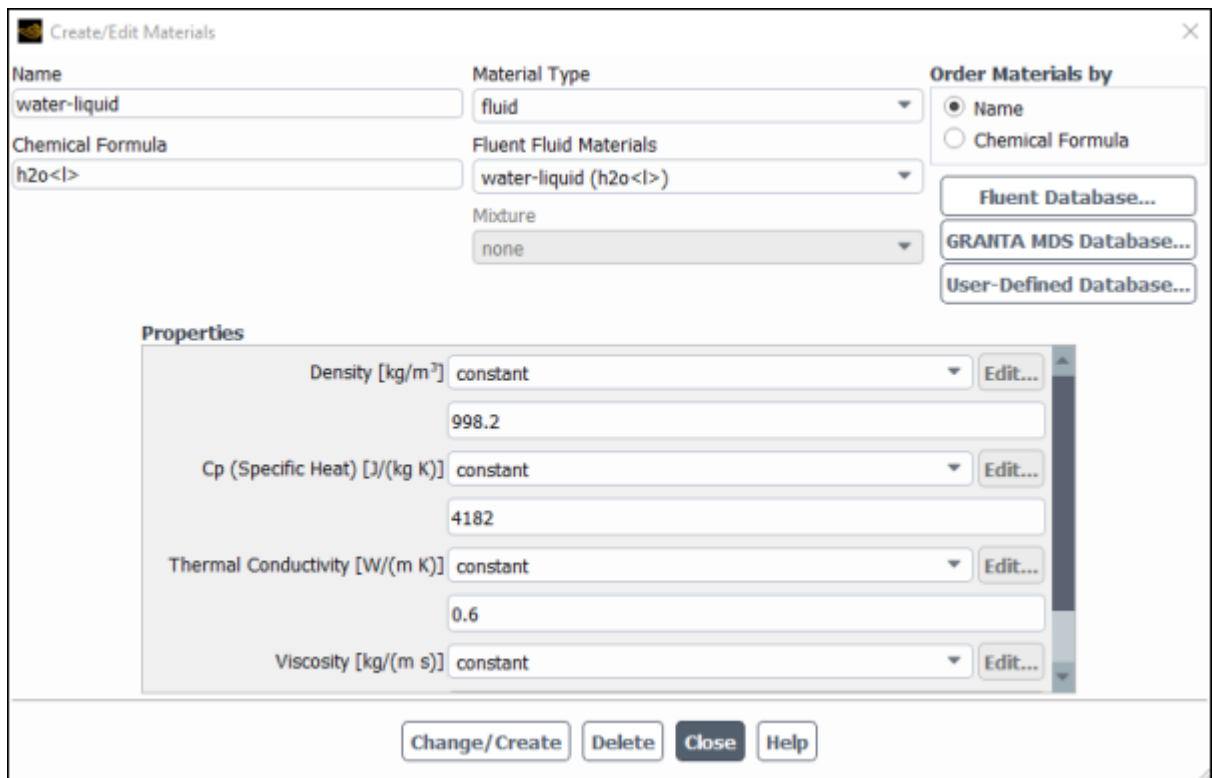
b. Copy the material **water-liquid (h2o < l >)** from the materials database (accessed by clicking the **Fluent Database...** button).

Figure 1.13: The Create/Edit Materials Dialog Box
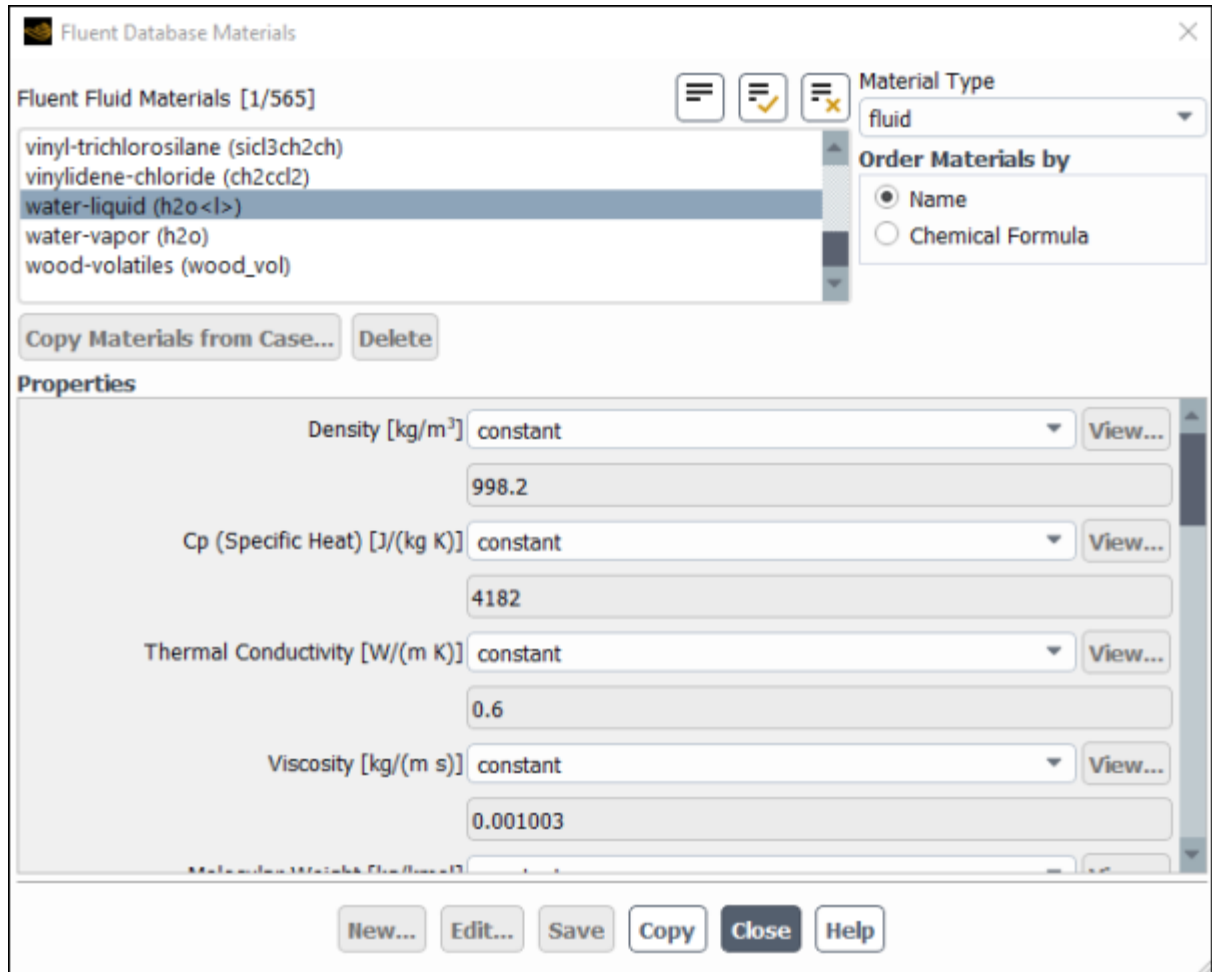
c. Select **water-liquid (h2o < l >)** from the materials list and Click **Copy**, then close the **Fluent Database...** panel.

d. Ensure that there are now two materials (water and air) defined locally by examining the **Fluent Fluid Materials** drop-down list in the **Create/Edit Materials** dialog box .

> **Note:**
>
> Both the materials will also be listed under **Fluid** in the **Materials** task page and under the **Materials** tree branch.

e. Close the **Create/Edit Materials** dialog box.

4. Set up the cell zone conditions for the CFD simulation using the **Zones** group of the **Physics** ribbon tab.

a. In the **Physics** tab, click **Cell Zones**.

 Physics → **Zones** → **Cell Zones**

This opens the **Cell Zone Conditions** task page.

Task Page

Cell Zone Conditions

Zone [Filter Text]

fluid

Phase Type ID

mixture ▼ fluid ▼ 4

Edit... Copy... Profiles...

Parameters... Operating Conditions...

Display Mesh...

Porous Formulation

◉ Superficial Velocity

◯ Physical Velocity
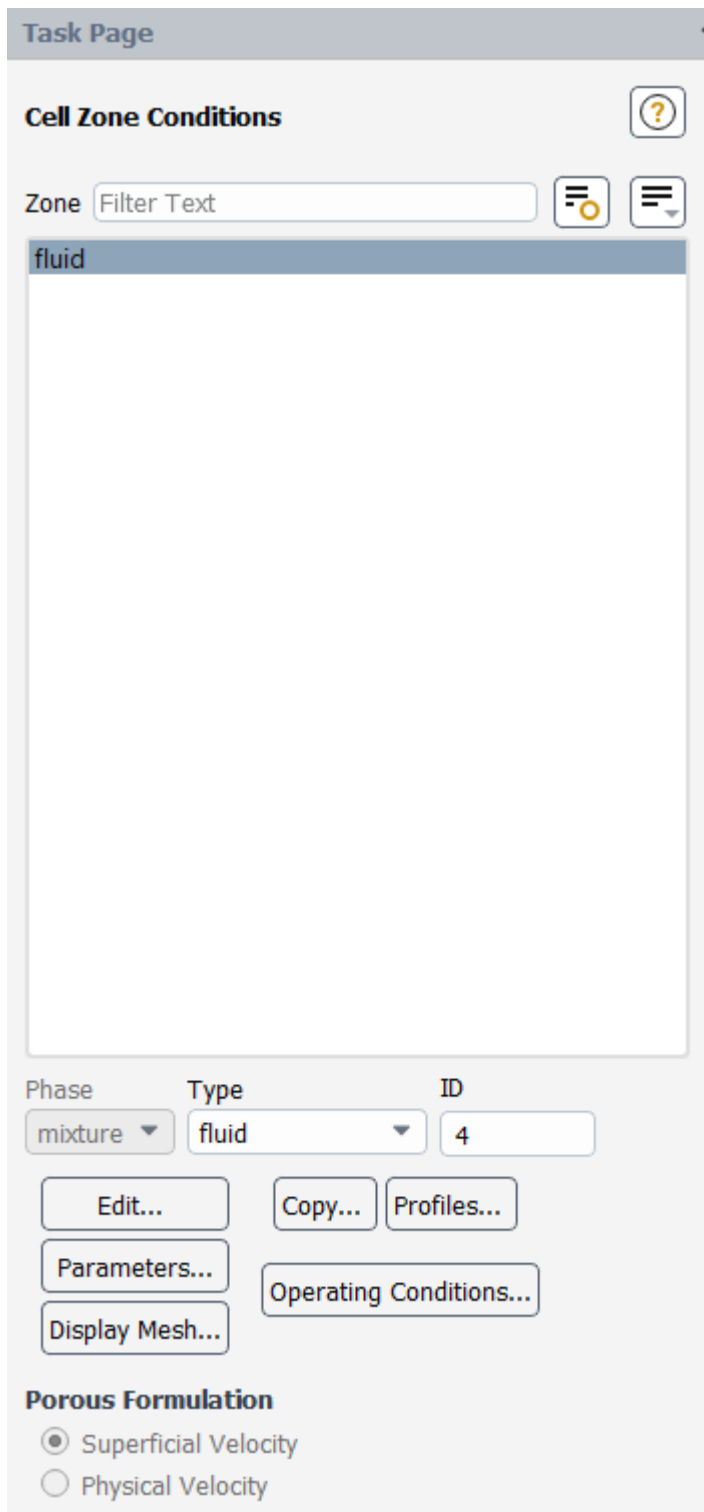
b. Set the cell zone conditions for the fluid zone.

i. In the **Cell Zone Conditions** task page, in the **Zone** list, select **fluid** and click **Edit...** to open the **Fluid** dialog box.

Note:

You can also double-click **fluid** in the **Cell Zones Conditions** task page or under the **Setup/Cell Zone Conditions** tree branch in order to open the corresponding dialog box.

---
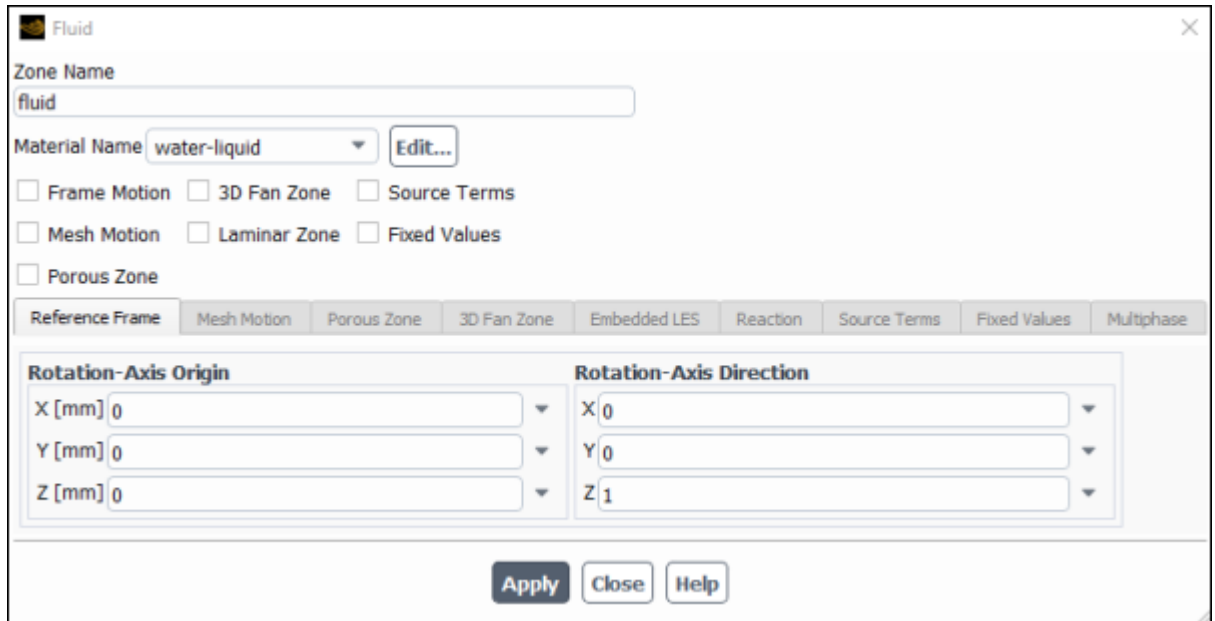
ii. Select **water-liquid** from the **Material Name** drop-down list.

iii. Click **Apply** and close the **Fluid** dialog box.

5. Set up the boundary conditions for the CFD analysis using the **Zones** group of the **Physics** ribbon tab.
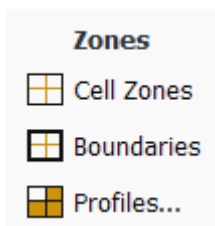
a. In the **Physics** tab, click **Boundaries** (**Zones** group).

 Physics → Zones → Boundaries

This opens the **Boundary Conditions** task page.
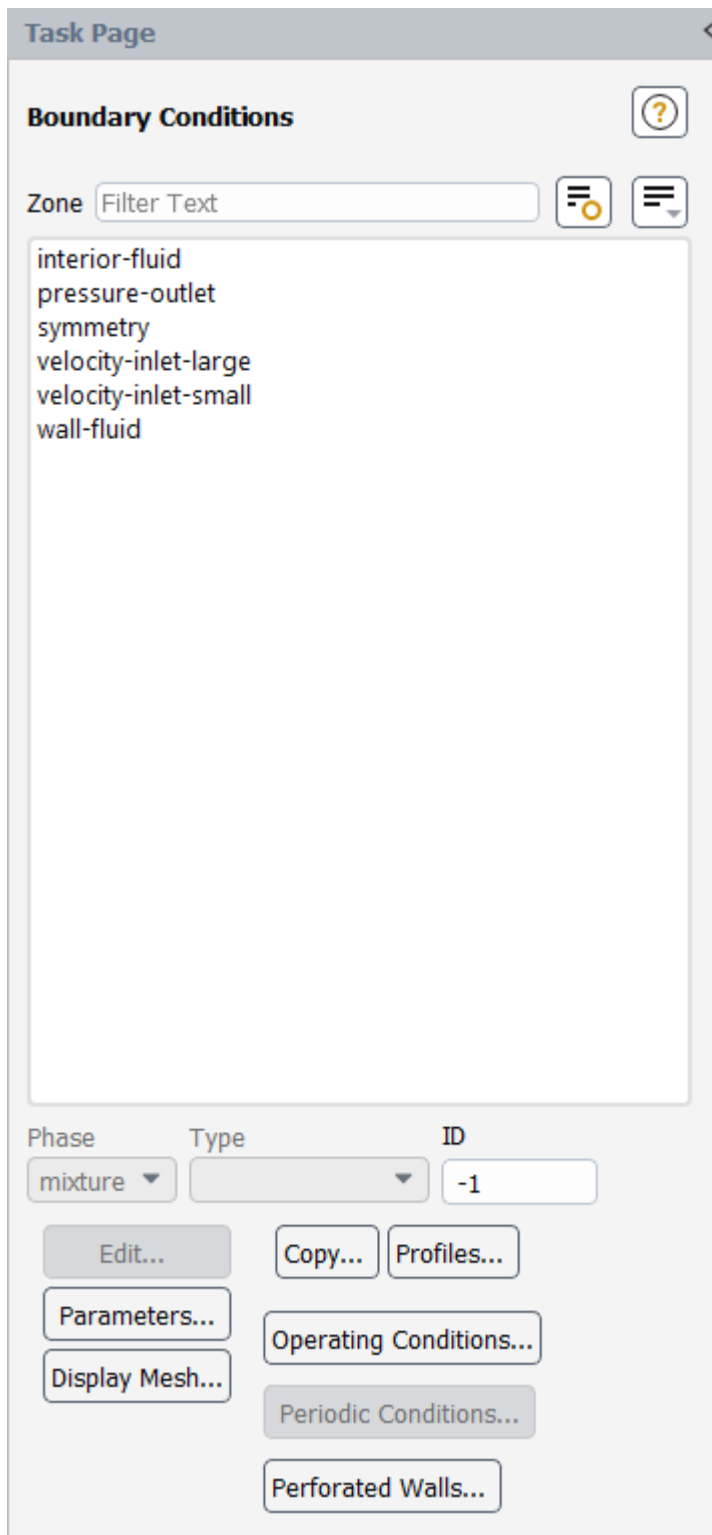
Note:

To display boundary zones grouped by zone type (as shown above), click the
Toggle Tree View button in the upper right corner of the **Boundary Conditions**
task page and under the **Group By** category select **Zone Type** from the drop-
down list.

b. Set the boundary conditions at the cold inlet (**velocity-inlet-large**).

Tip:

If you are unsure of which inlet zone corresponds to the cold inlet, you can use
the mouse to probe for mesh information in the graphics window. If you click
the right mouse button with the pointer on any node in the mesh, information
about the associated zone will be displayed in the Ansys Fluent console, including
the name of the zone. The zone you probed will be automatically selected from
the **Zone** selection list in the **Boundary Conditions** task page.

Alternatively, you can click the probe button (⬚) in the graphics toolbar and
click the left mouse button on any node. This feature is especially useful when
you have several zones of the same type and you want to distinguish between
them quickly. The information will be displayed in the console.

---

i. From the **Zone** selection list, select **velocity-inlet-large** and click **Edit...**.
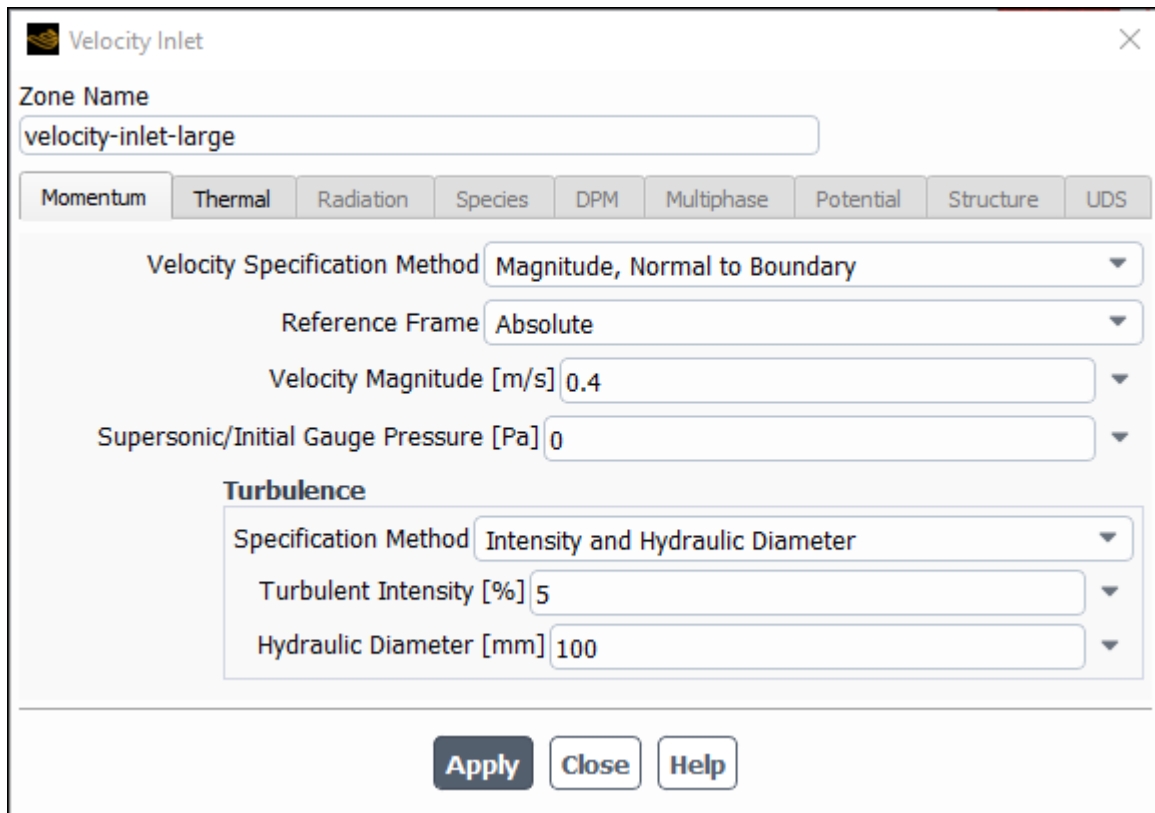
ii. In the **Velocity Inlet** dialog box, ensure that **Magnitude, Normal to Boundary** is selected
for **Velocity Specification Method**.

iii. Enter $0.4\ m/s$ for **Velocity Magnitude**.

iv. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity
and Hydraulic Diameter**.

---

v. Retain the default of 5 % for **Turbulent Intensity**.

vi. Enter `100` mm for **Hydraulic Diameter**.

Note:

The hydraulic diameter D_h is defined as:

$$D_h = \frac{4A}{P_w}$$

where A is the cross-sectional area and P_w is the wetted perimeter.

---

vii. Click the **Thermal** tab.
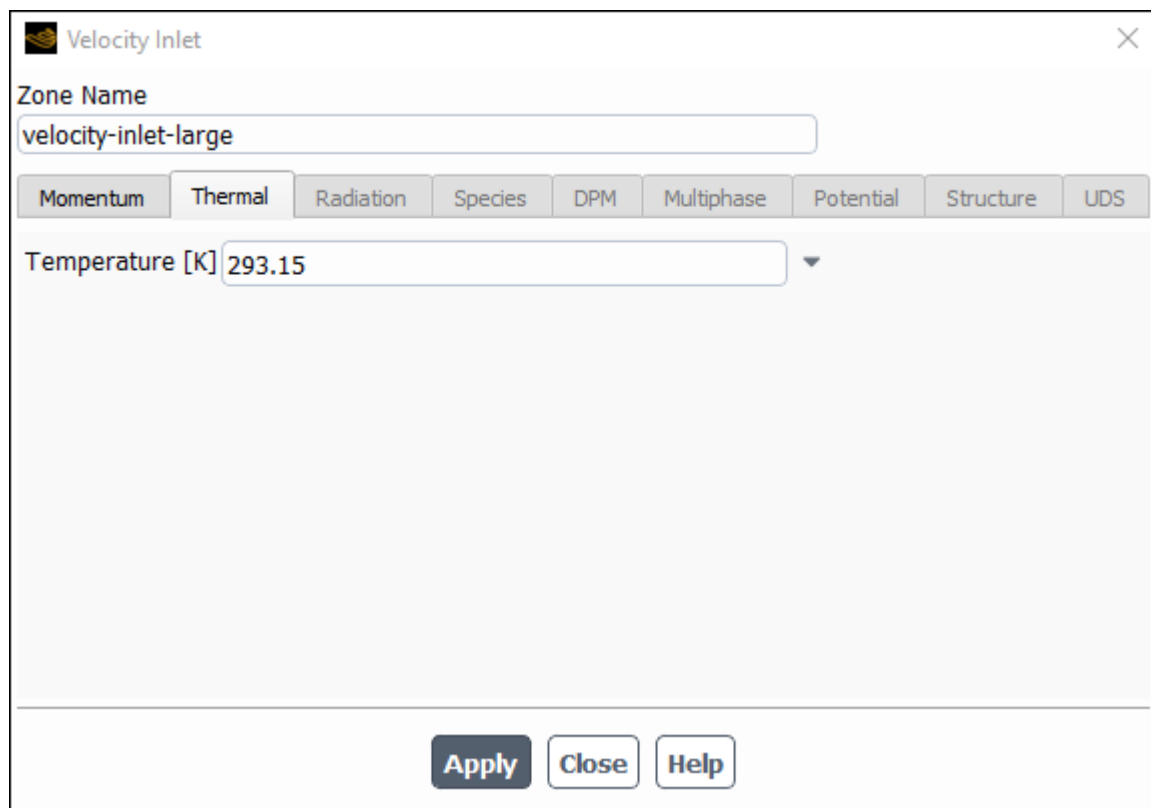
viii. Enter `293.15` K for **Temperature**.

ix. Click **Apply** and close the **Velocity Inlet** dialog box.

c. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-small**), using the values in the following table:

Setting	Value
Velocity Specification Method	Magnitude, Normal to Boundary
Velocity Magnitude	`1.2` [m/s]

Setting	Value
Specification Method	Intensity & Hydraulic Diameter
Turbulent Intensity	5%
Hydraulic Diameter	**25** [mm]
Temperature	**313.15**[K]

d. Double-click **pressure-outlet** and set the boundary conditions at the outlet, as shown in the **Pressure Outlet** dialog box.
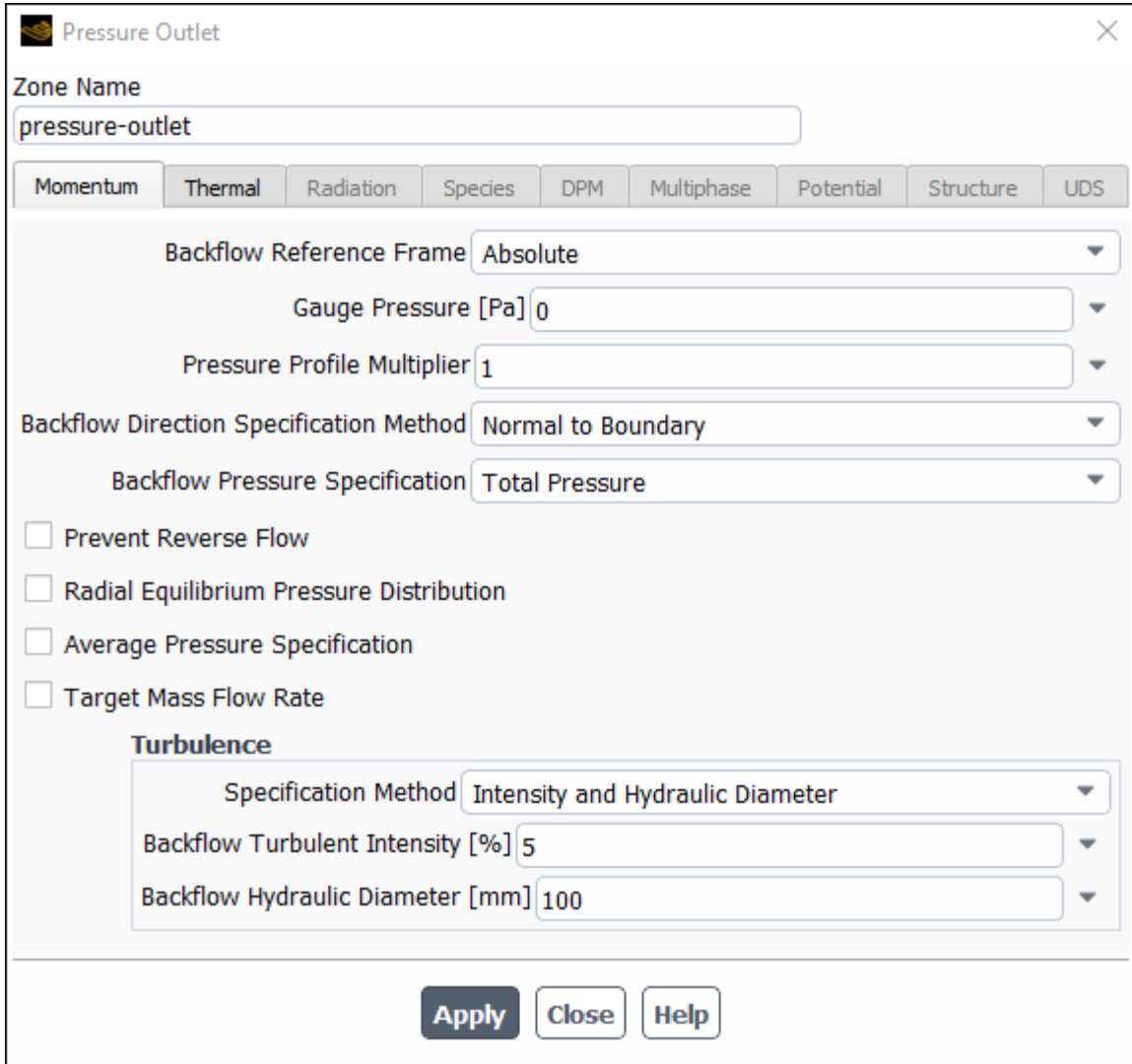
Note:

Ansys Fluent will use the backflow conditions only if the fluid is flowing into the computational domain through the outlet. Since backflow might occur at some point during the solution procedure, you should set reasonable backflow conditions to prevent convergence from being adversely affected.

1.4.6. Solving

In the steps that follow, you will set up and run the calculation using the **Solution** ribbon tab.

> **Note:**
>
> You can also use the task pages listed under the **Solution** branch in the tree to perform solution-related activities.

1. Set up solution parameters for the CFD simulation.

    a. i. In the **Solution** ribbon tab, click **Methods...** (**Solution** group).

   Solution → Solution → Methods...

   This will open the **Solution Methods** task page.
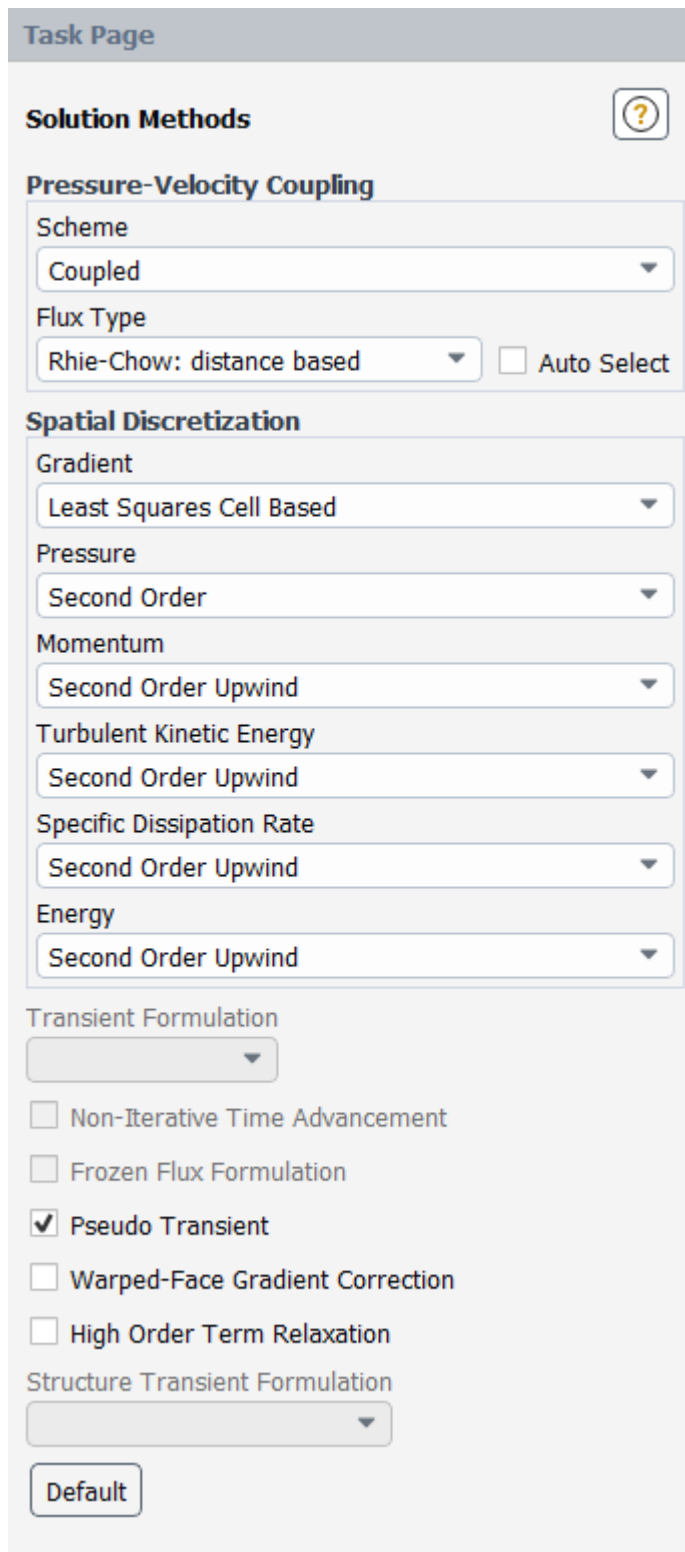
  ii. Retain the default selections.

 b. Enable the plotting of residuals during the calculation.

  i. In the **Solution** ribbon tab, click **Residuals...** (**Reports** group).

![Solution icon] **Solution → Reports → Residuals...**

Reports
🗠 Residuals...   ✳ Convergence...
📊 Definitions ▾   📄 File...   📈 Plot...

Residual Monitors   ✕

Options   **Equations**

☑ Print to Console   Residual   Monitor   Check Convergence Absolute Criteria

☑ Plot   continuity   ☑   ☑   0.001

  [Curves...] [Axes...] x-velocity   ☑   ☑   0.001

  Iterations to Plot   y-velocity   ☑   ☑   0.001

  [1000   ▲▼]   z-velocity   ☑   ☑   0.001

                         energy   ☑   ☑   1e-06

Iterations to Store   k   ☑   ☑   0.001

[1000   ▲▼]   omega   ☑   ☑   0.001

                        [Convergence Conditions...]

                        ☐ Show Advanced Options

              [OK] [Plot] [Cancel] [Help]

In the **Residual Monitors** dialog box, ensure that **Plot** is enabled in the **Options** group.

ii. Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.

iii. Click **OK** to close the **Residual Monitors** dialog box.

Note:

By default, all variables will be monitored and checked by Ansys Fluent as a means to determine the convergence of the solution.

---

c. Create a surface report definition at the outlet (**pressure-outlet**).

![Solution icon] **Solution → Reports → Definitions → New → Surface Report → Facet Maximum**
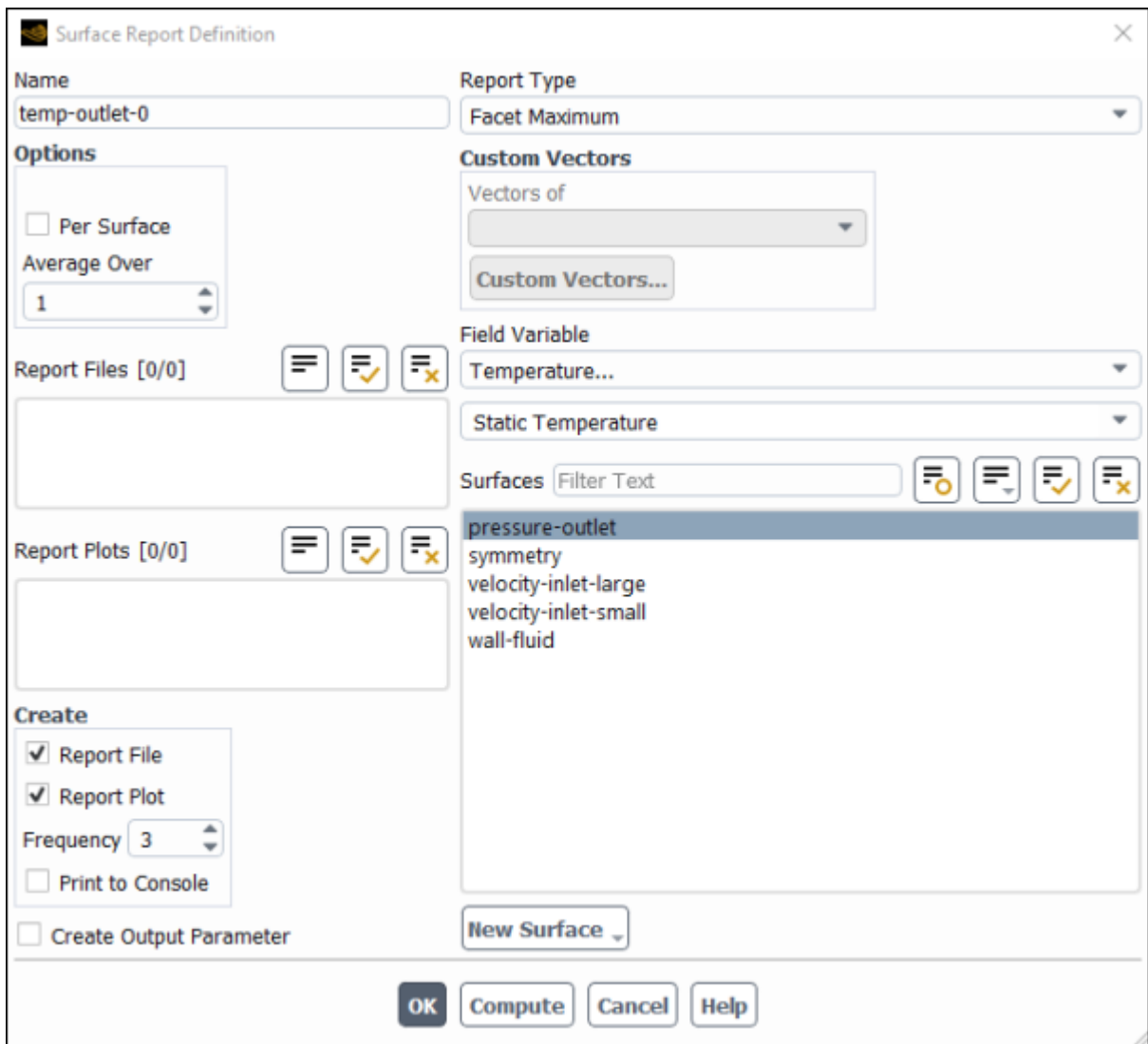
i.   In the **Surface Report Definition** dialog box, enter `temp-outlet-0` for the **Name**.

ii. Under the **Create** group, enable **Report File** and **Report Plot**.

     During a solution run, Ansys Fluent will write solution convergence data in a report file
     and plot the solution convergence history in a graphics window.

     *It is good practice to monitor physical solution quantities in addition to equation residuals
     when assessing convergence.*

iii. Set **Frequency** to 3 by clicking the up-arrow button.

     *This setting instructs Ansys Fluent to update the plot of the surface report and write data to a
     file after every 3 iterations during the solution.*

iv. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

v.   Select **pressure-outlet** from the **Surfaces** selection list.

vi. Click **OK** to save the surface report definition and close the **Surface Report Definition** dialog box.

The new surface report definition `temp-outlet-0` appears under the **Solution/Report Definitions** tree item. Ansys Fluent also automatically creates the following items:

- `temp-outlet-0-rfile` (under the **Solution/Monitors/Report Files** tree branch)

- `temp-outlet-0-rplot` (under the **Solution/Monitors/Report Plots** tree branch)

d. In the tree, double-click the **temp-outlet-0-rfile** (under the **Solution/Monitors/Report Files**) and examine the report file settings in the **Edit Report File** dialog box.
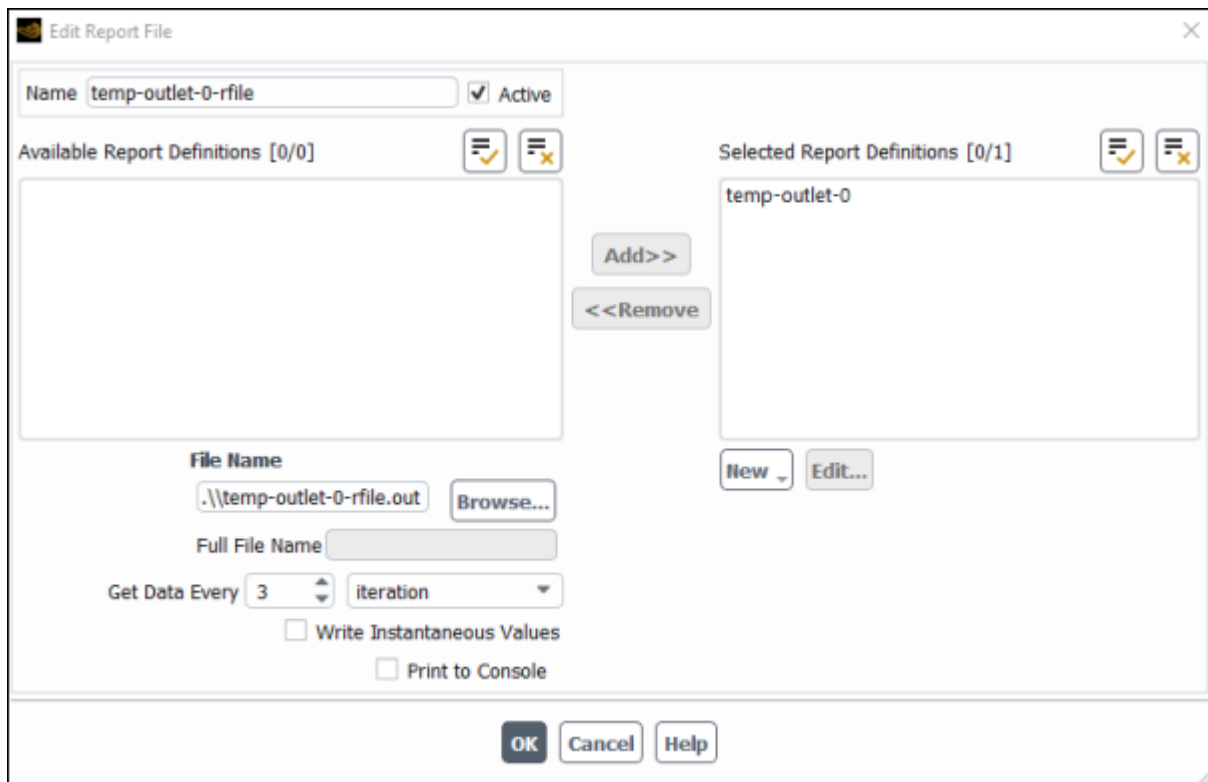
The dialog box is automatically populated with data from the `temp-outlet-0` report definition. The report that will be written in a report file during a solution is listed under **Selected Report Definitions**.

Retain the default output file name and click **OK**.

e. In the tree, double-click **temp-outlet-0-rplot** (under **Solution/Monitors/Report Plots**) and examine the report plot settings in the **Edit Report Plot** dialog box.
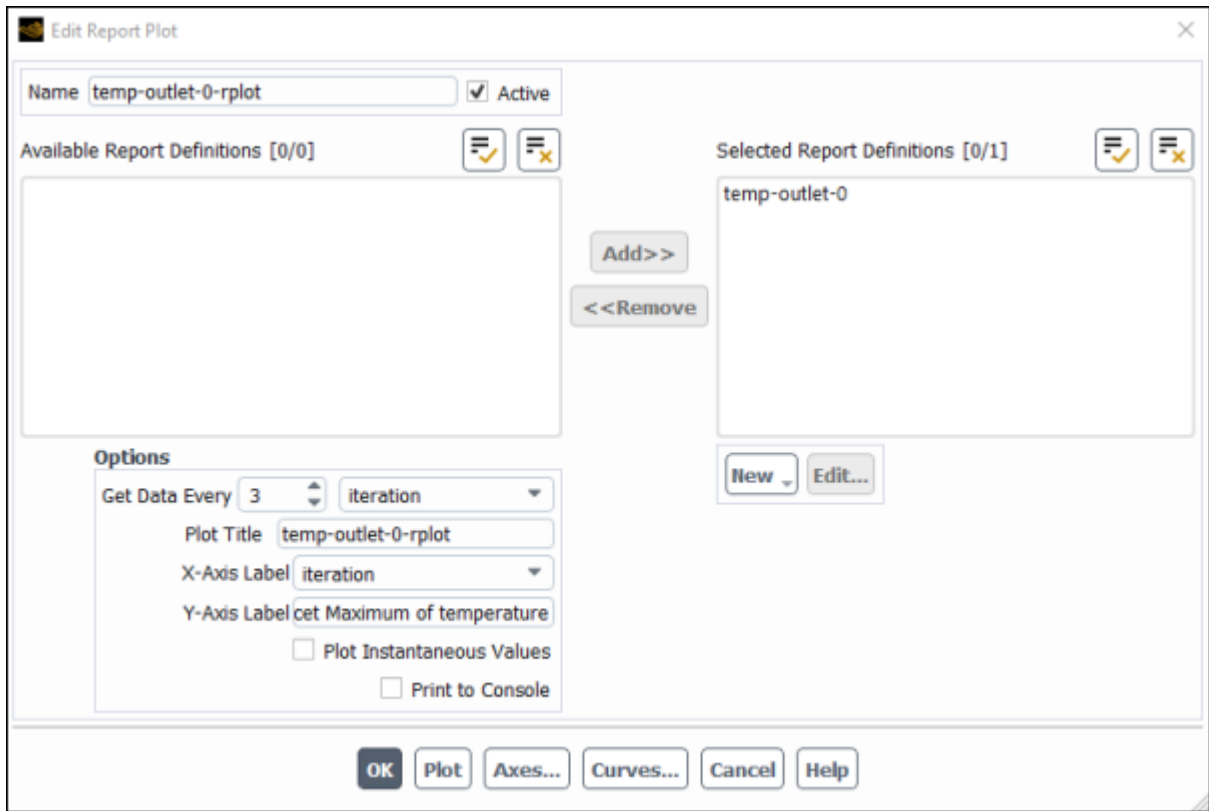
The dialog box is automatically populated with data from the `temp-outlet-0` report
definition. As the solution progresses, the report that is listed under **Selected Report Defini-
tions** will be plotted in a graphics tab window with the title specified in **Plot Title**.

Retain the default names for **Plot Title** and **Y-Axis Label** and click **OK**.

Note:

You can create report definitions for different boundaries and plot them in the same
graphics window. However, report definitions in the same **Report Plot** must have
the same units.

---

f.   Initialize the flow field using the **Initialization** group of the **Solution** ribbon tab.

 Solution → Initialization
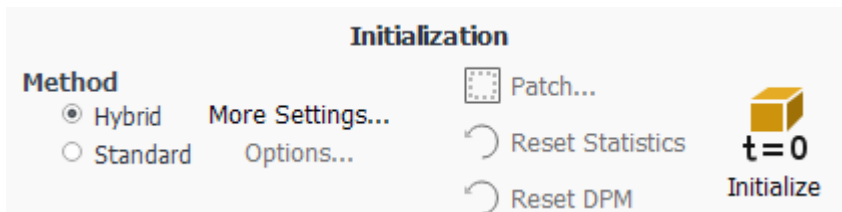
i.   Keep the **Method** at the default of **Hybrid**.

ii. Click **Initialize**.

---

2. Calculate a solution using the **Run Calculation** group of the **Solution** tab.

▦ **Solution → Run Calculation**

Run Calculation

☑ Input Summary...   ↗ Run Calculation...

☑ Check Case   Time Scale Factor [1]   ▾

↯ Update Dynamic Mesh...   No. of Iterations [250] ⌃⌄
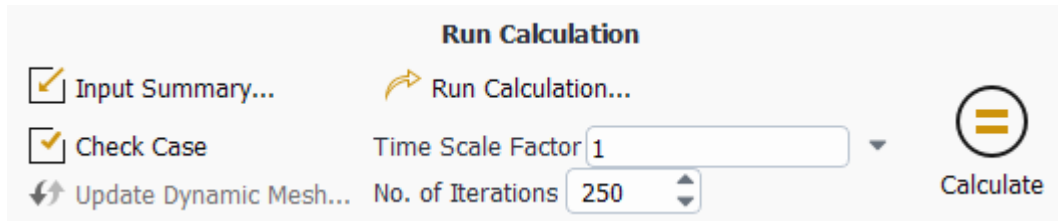
⊜ Calculate

a. Start the calculation by requesting 250 iterations.

  i. Enter 250 for **No. of Iterations**.

  ii. Click **Calculate**.

  Note that while the program is calculating the solution, the states of the **Setup** and **Solution** cells in the fluid flow Ansys Fluent analysis system in Ansys Workbench are changing. For example:

  • The state of the **Setup** cell becomes **Up-to-Date** and the state of the **Solution** cell becomes **Refresh Required** after the **Run Calculation** task page is visited and the number of iterations is specified.

  • The state of the **Solution** cell is **Update Required** while iterations are taking place.

  • The state of the **Solution** cell is **Up-to-Date** when the specified number of iterations are complete (or if convergence is reached).

  As the calculation progresses, the surface report plot will be plotted in the graphics window (**temp-outlet-0–rplot** tab window) (Figure 1.14: Convergence History of the Maximum Temperature at Pressure Outlet (p. 44)).

Figure 1.14: Convergence History of the Maximum Temperature at Pressure Outlet
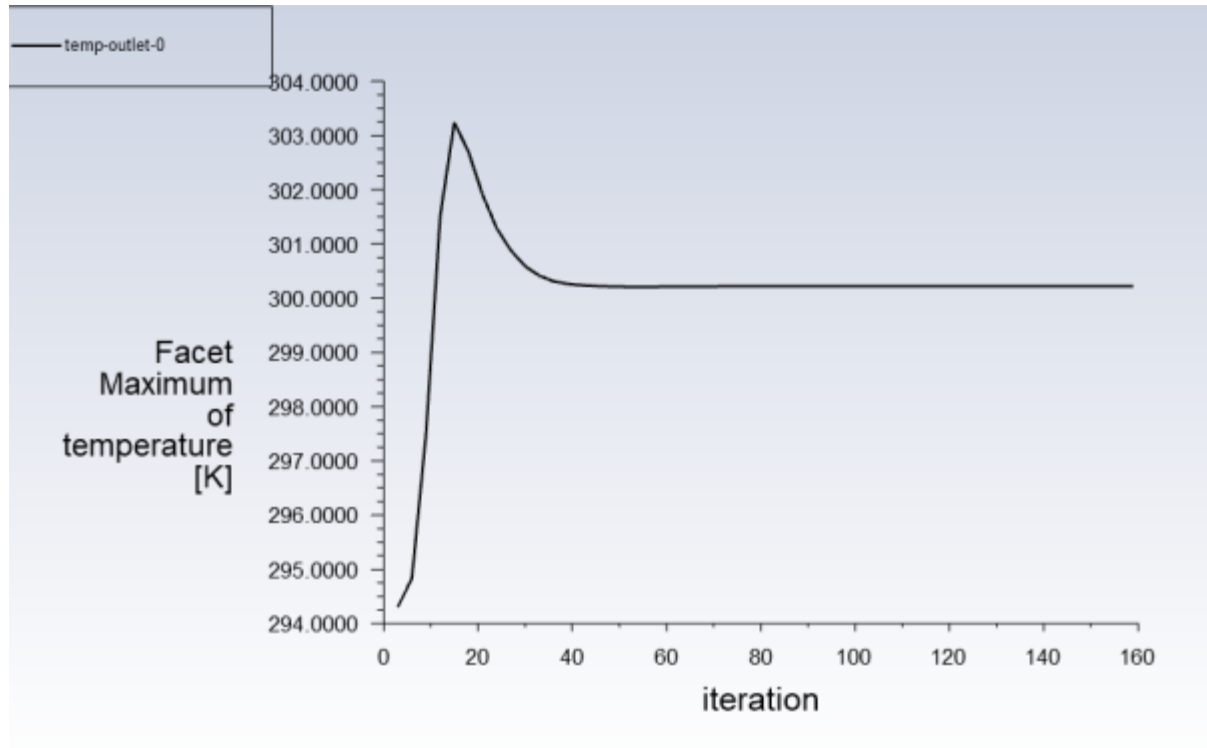
The residuals history will be plotted in the **Scaled Residuals** tab window (Figure 1.15: Residuals for the Converged Solution (p. 45)).

Figure 1.15: Residuals for the Converged Solution
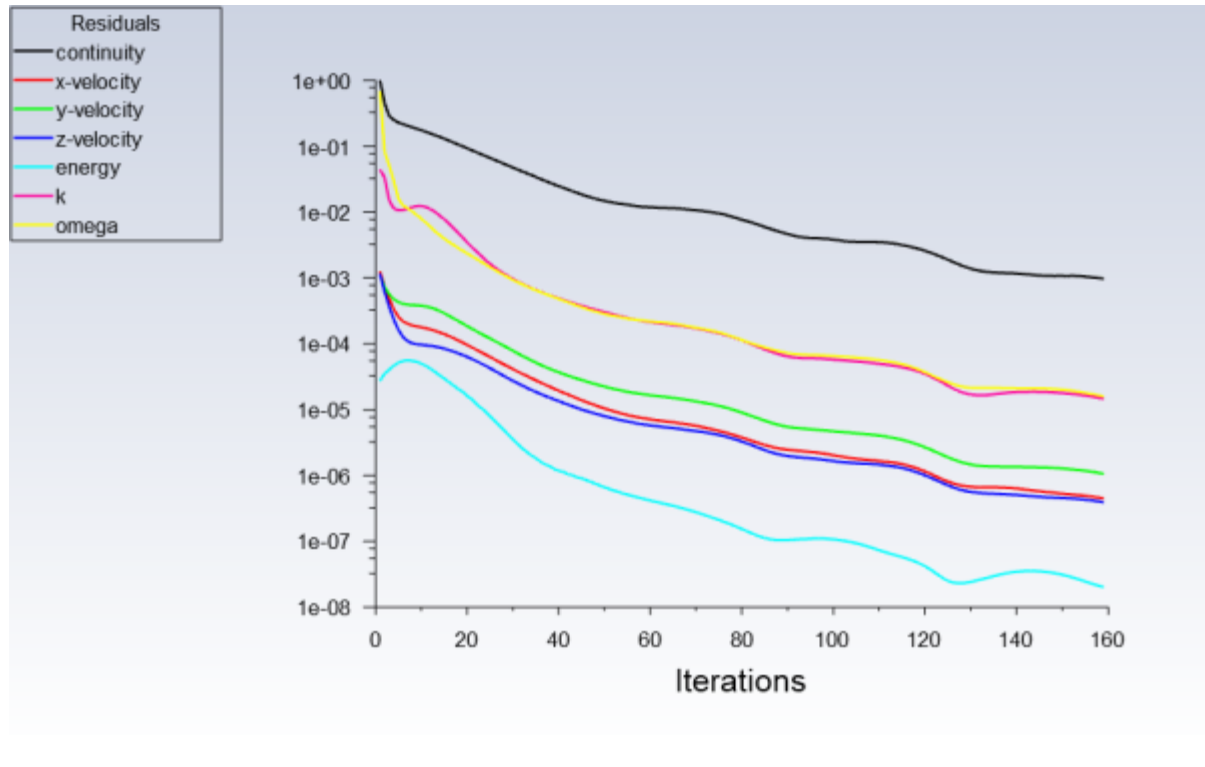
Note:

The solution will be stopped by Ansys Fluent when the residuals reach their specified values or after 250 iterations. The exact number of iterations will vary depending on the platform being used. An **Information** dialog box will open to alert you that the calculation is complete. Click **OK** in the **Information** dialog box to proceed.

Because the residual values vary slightly by platform, the plot that appears on your screen may not be exactly the same as the one shown here.

Note that the Ansys Fluent settings file (`FFF.set`) is updated in the **Files** view of the Ansys Workbench before the calculation begins.

b. Examine the plots for convergence (Figure 1.15: Residuals for the Converged Solution (p. 45) and Figure 1.14: Convergence History of the Maximum Temperature at Pressure Outlet (p. 44)).

Note:

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

There are three indicators that convergence has been reached:

- The residuals have decreased to a sufficient degree.

The solution has converged when the **Convergence Criterion** for each variable has been reached. The default criterion is that each residual will be reduced to a value of less than 10^{-3}, except the **energy** residual, for which the default criterion is 10^{-6}.

- The solution no longer changes with more iterations.

  Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

- The overall mass, momentum, energy, and scalar balances are obtained.

  You can examine the overall mass, momentum, energy and scalar balances in the **Flux Reports** dialog box. The net imbalance should be less than 0.2 % of the net flux through the domain when the solution has converged. In the next step you will check to see if the mass balance indicates convergence.

3. With Ansys Fluent still running, go back to Ansys Workbench and view the list of generated files.

   View → **Files**

   Note the addition of the surface report definition file `temp-outlet-0.out` to the list of files.

   The status of the **Solution** cell is now up-to-date.
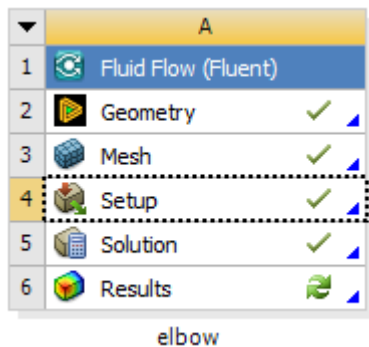
1.4.7. Displaying Results in Ansys Fluent and CFD-Post

In this step, you will display the results of the simulation in Ansys Fluent, display the results in CFD-Post, and then review the list of files generated by Ansys Workbench.

1. Display results in Ansys Fluent using the **Results** ribbon tab.

   In Ansys Fluent, you can perform a simple evaluation of the velocity and temperature contours on the symmetry plane. Later, you will use CFD-Post (from within Ansys Workbench) to perform the same evaluation.

a. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.16: Velocity Distribution Along Symmetry Plane (p. 48)).

Results → Graphics → Contours → New...

Note:

You can also double-click the **Results/Graphics/Contours** tree item.

---
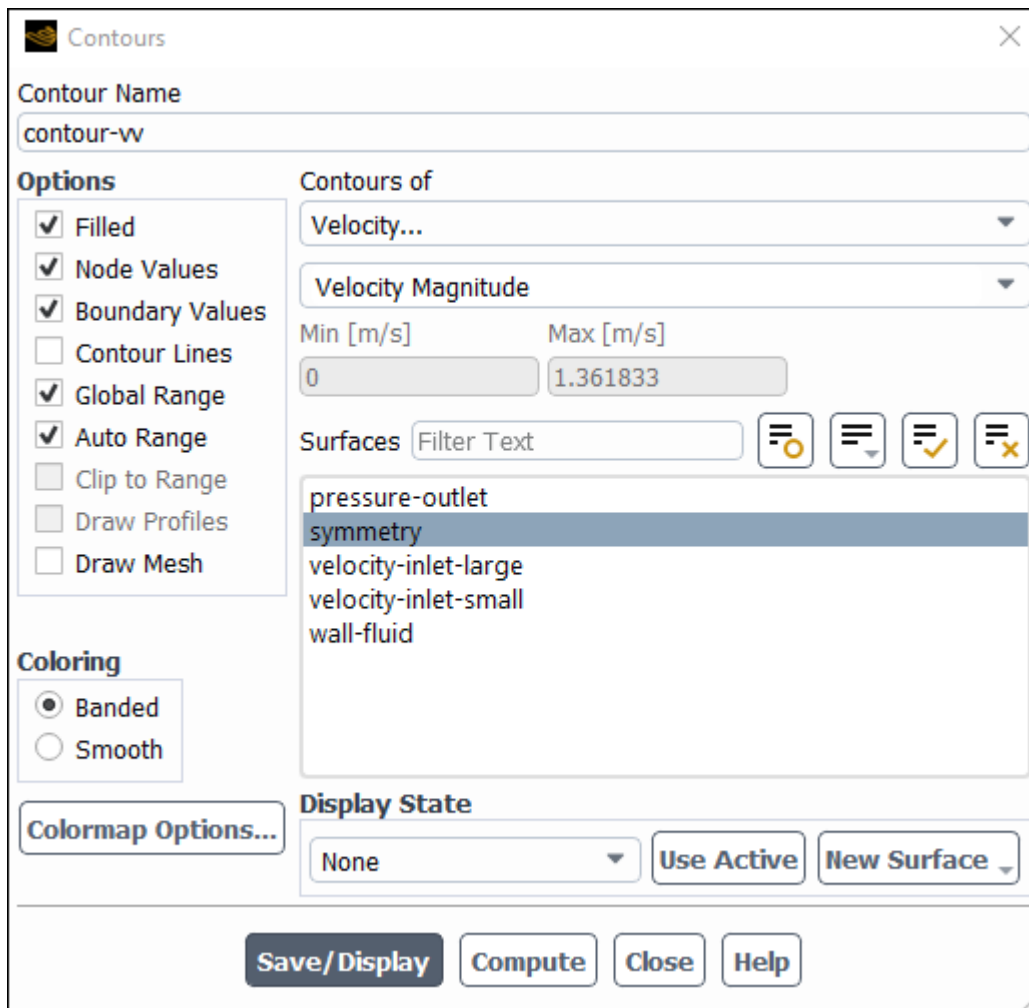
i. Enter `contour-vv` for **Contour Name**.

ii. In the **Contours** dialog box, in the **Options** group, enable **Filled**.

iii. Select **Banded** in the **Coloring** group box.

iv. Ensure that **Node Values** and **Boundary Values** is enabled in the **Options** group.

v. From the **Contours of** drop-down lists, select **Velocity...** and **Velocity Magnitude**.

vi. From the **Surfaces** selection list, deselect all items by clicking ⊠ and then select **symmetry**.

vii. Click **Display** to display the contours in the active graphics window.

Figure 1.16: Velocity Distribution Along Symmetry Plane
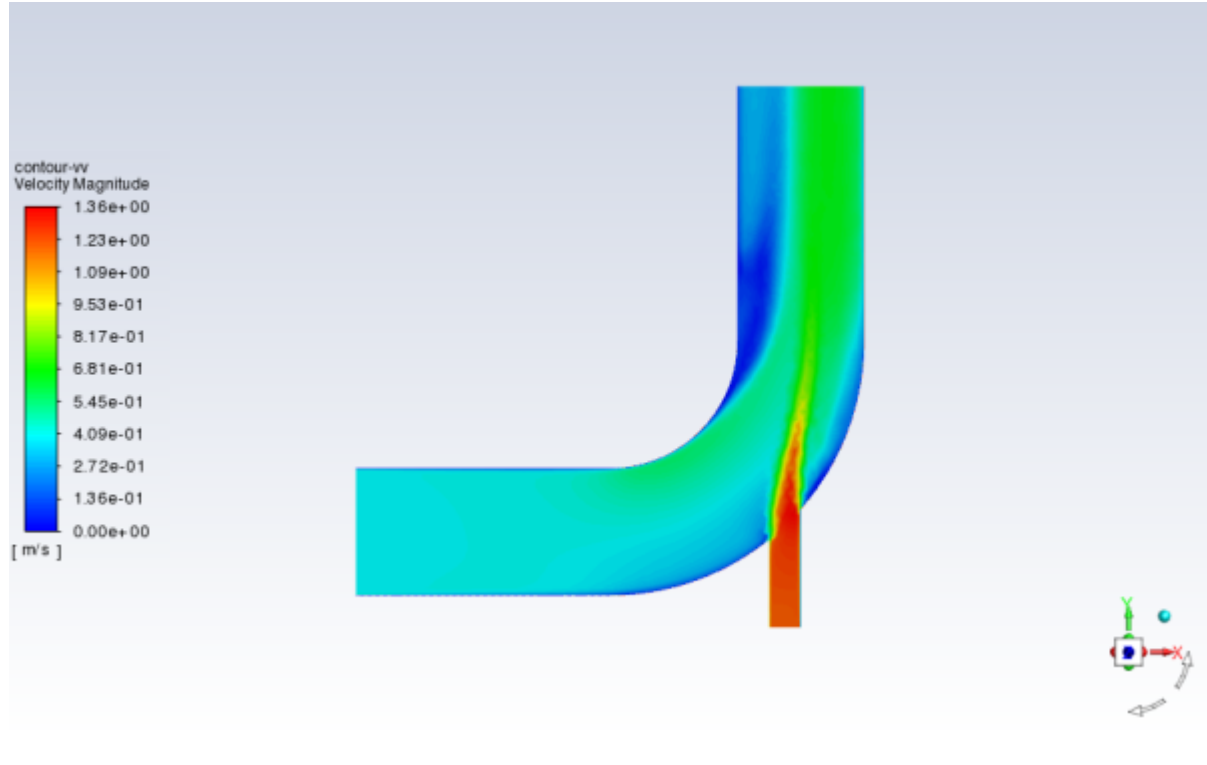
Note:

You may want to clear **Headlight** and **Lighting** in the **Viewing** ribbon tab (**Display** group).

---

b. Display filled contours of temperature on the symmetry plane (Figure 1.17: Temperature Distribution Along Symmetry Plane (p. 50)).

Results → **Graphics** → **Contours** → **New...**
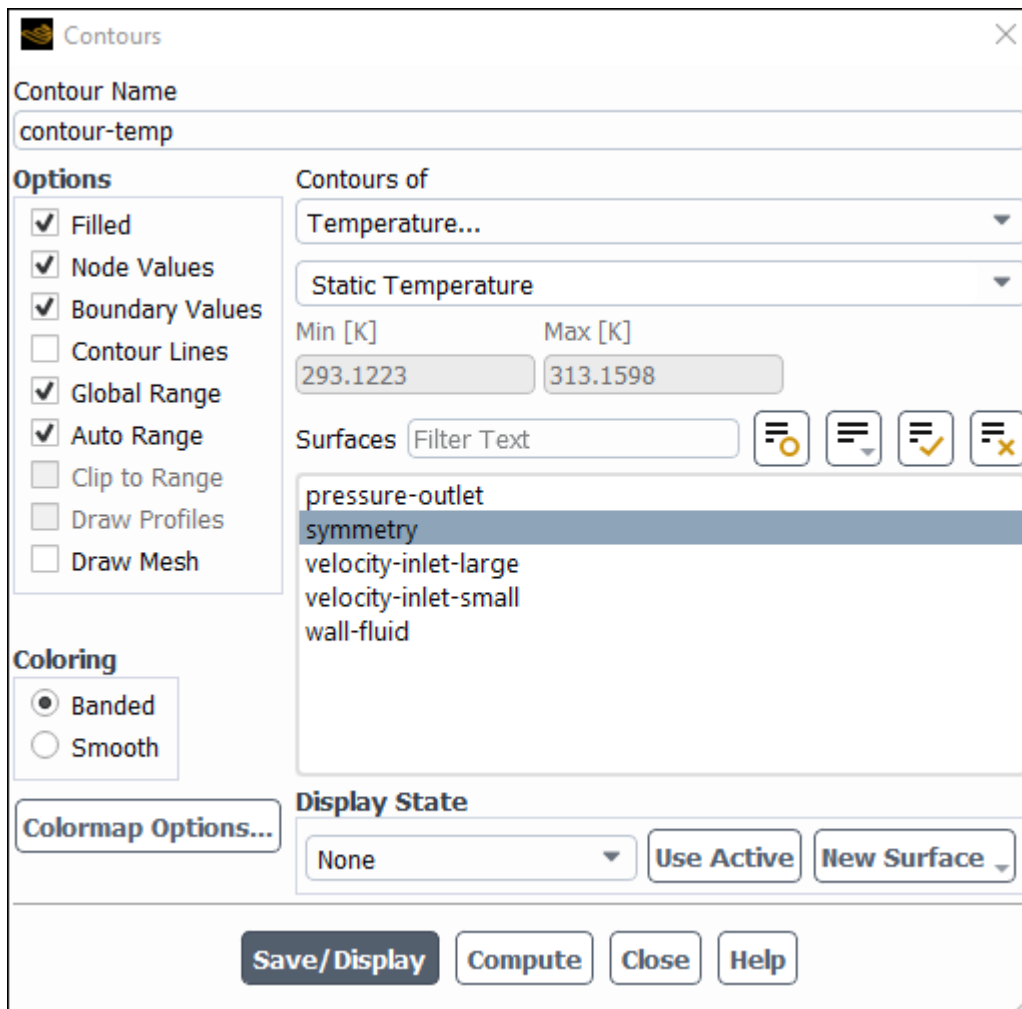
i. Enter `contour-temp` for **Contour Name**.

ii. Select **Banded** in the **Coloring** group box.

iii. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

iv. Click **Display** and close the **Contours** dialog box.

Figure 1.17: Temperature Distribution Along Symmetry Plane
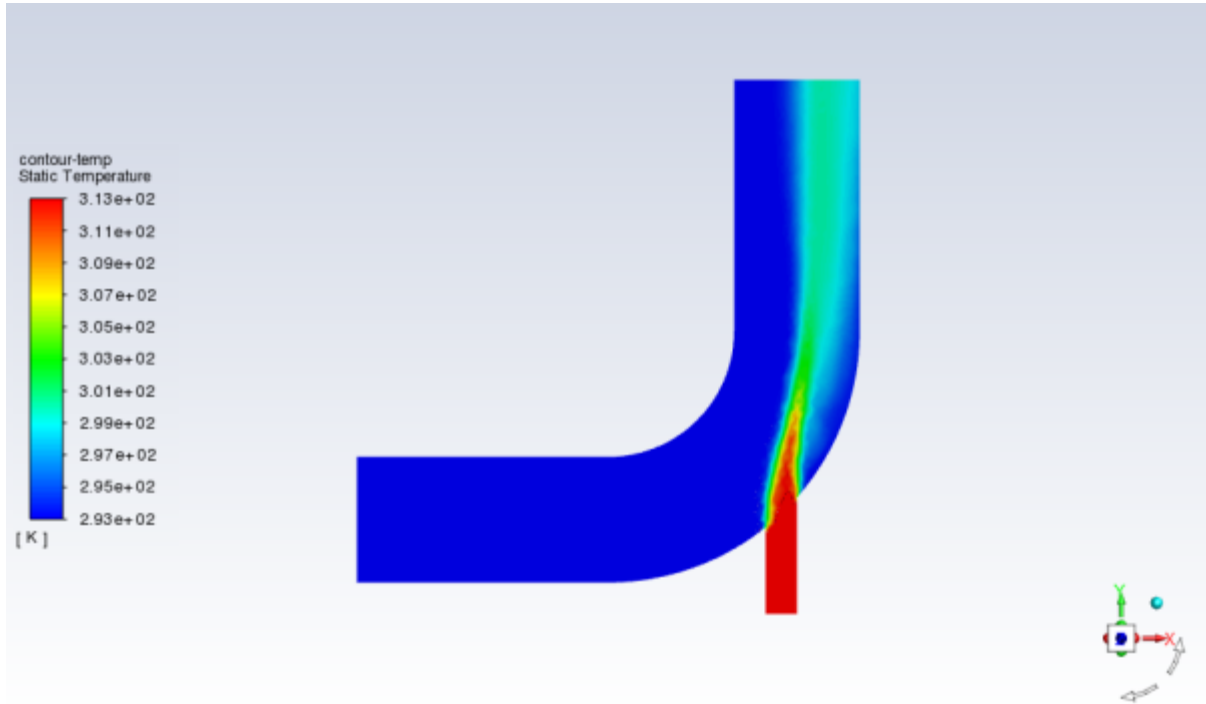
c. Close the Ansys Fluent application.

File → Close Fluent

> **Important:**
>
> Note that the Ansys Fluent case and data files are automatically saved when you exit Ansys Fluent and return to Ansys Workbench.
>
> The following prompt is presented, whenever settings change in Ansys Fluent and in the **Project Schematic** the **Solution** cell might need to be refreshed..
>
>
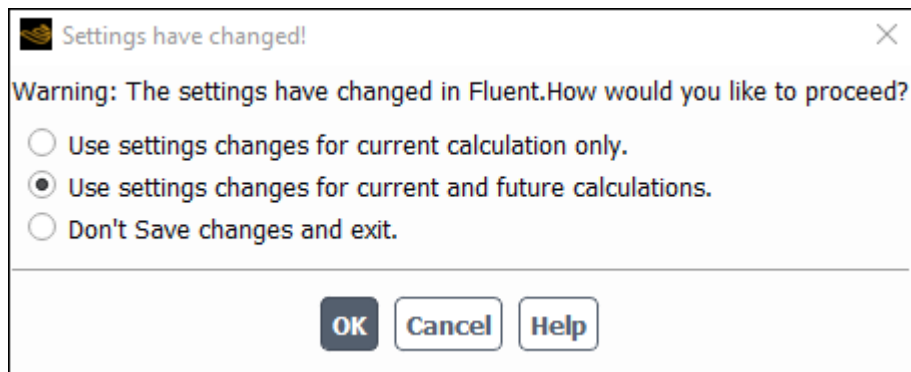
d. View the list of files generated by Ansys Workbench.

View → Files

Note the addition of the compressed Ansys Fluent case and data files to the list of files. These will have names like `FFF-1.cas.gz` and `FFF-1-00167.dat.gz`. Note that the digit(s) following `FFF` may be different if you have had to restart the meshing or calculation steps for any reason and that the name of the data file is based on the number of iterations. Thus your file names may be slightly different than those shown here.

2. Display results in CFD-Post.

   a. Start CFD-Post.

      In the Ansys Workbench **Project Schematic**, double-click the **Results** cell in the `elbow` fluid flow analysis system (cell `A6`). This displays the CFD-Post application. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

      > **Note:**
      >
      > The elbow geometry is already loaded and is displayed in outline mode. Ansys Fluent case and data files are also automatically loaded into CFD-Post.

      Figure 1.18: The Elbow Geometry Loaded into CFD-Post
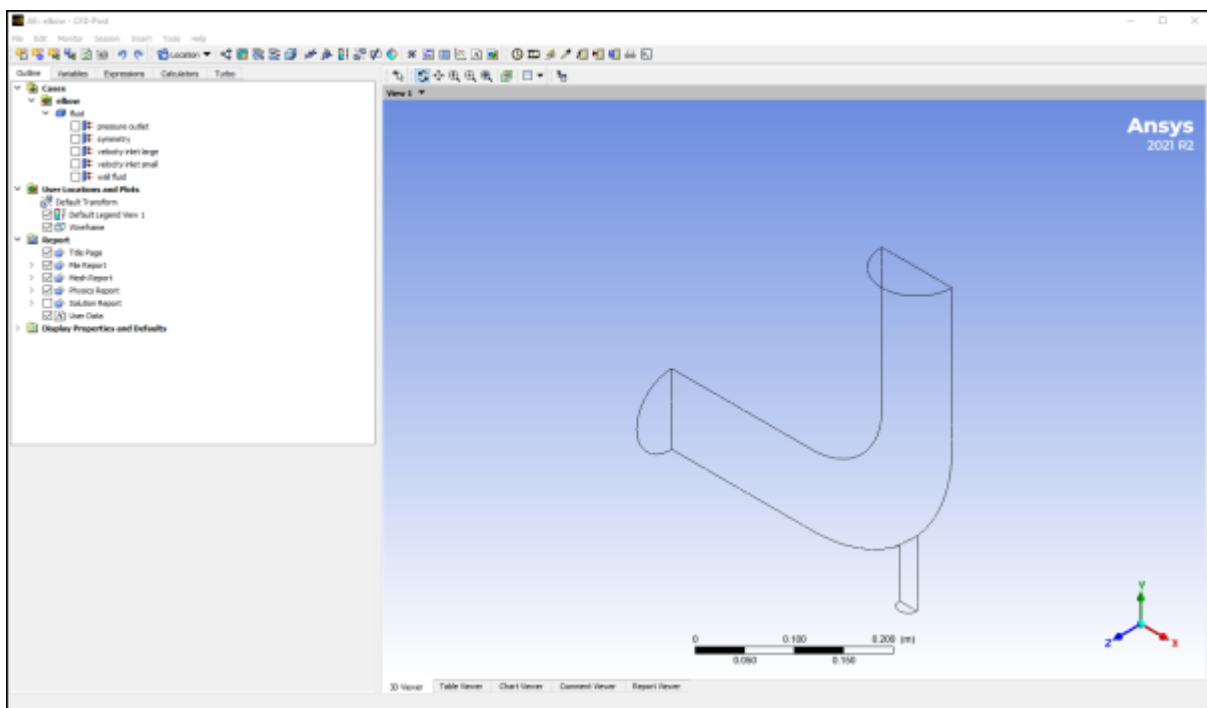
   b. Reorient the display.

      Click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.

   c. Ensure that *Highlighting (* *)* is disabled.

d. Display contours of velocity magnitude on the symmetry plane (Figure 1.19: Velocity Distribution Along Symmetry Plane (p. 53)).

i. Insert a contour object using the **Insert** menu item at the top of the CFD-Post window.

Insert → Contour

This displays the **Insert Contour** dialog box.

ii. Keep the default name of the contour (`Contour 1`) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.

iii. In the **Geometry** tab, from the **Domains** drop-down list, select **fluid**.

iv. In the **Locations** list, select **symmetry**.

v. In the **Variable** list, select **Velocity**.

vi. Click **Apply**.

Details of **Contour 1**

| Geometry | Labels | Render | View |

Domains: fluid

Locations: symmetry

Variable: Velocity

Range: Global

Min: 0 [m s^-1]

Max: 1.358 [m s^-1]

# of Contours: 11

Advanced Properties

Apply   Reset   Defaults
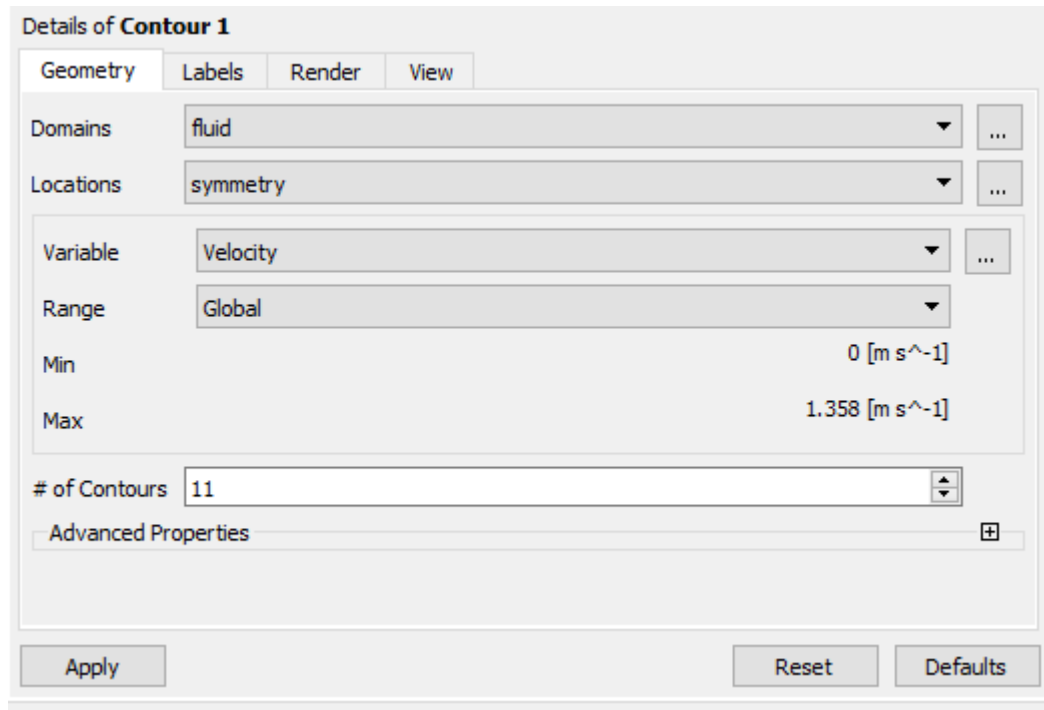
Figure 1.19: Velocity Distribution Along Symmetry Plane
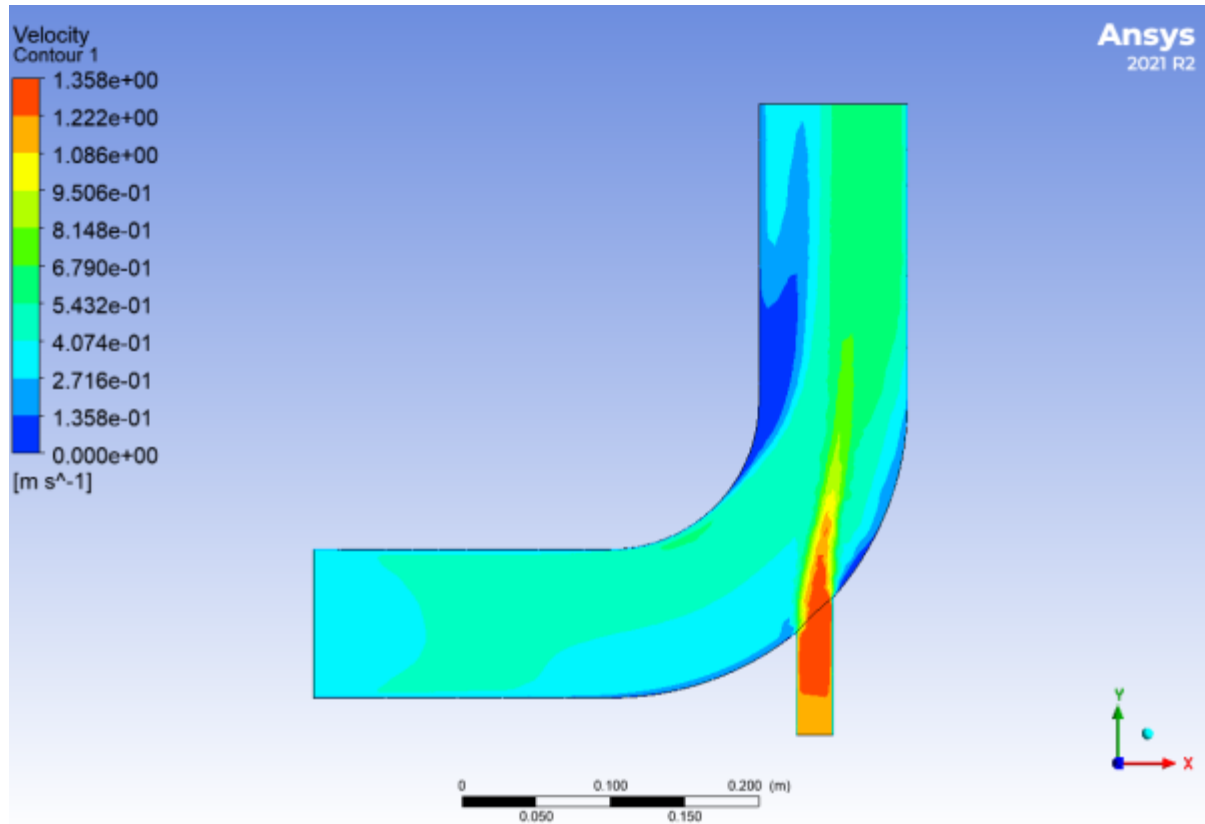
e. Display contours of temperature on the symmetry plane (Figure 1.20: Temperature Distribution Along Symmetry Plane (p. 54)).

i. In the **Outline** tree view, under **User Locations and Plots**, clear the check box beside the **Contour 1** object to disable the **Contour 1** object and hide the first contour display.

ii. Insert a contour object.

Insert → Contour

This displays the **Insert Contour** dialog box.

iii. Keep the default name of the contour (`Contour 2`) and click **OK** to close the dialog box. This displays the **Details of Contour 2** view below the **Outline** view.

iv. In the **Geometry** tab, from the **Domains** drop-down list, select **fluid**.

v. In the **Locations** list, select **symmetry**.

vi. In the **Variable** list, select **Temperature**.

vii. Click **Apply**.

Figure 1.20: Temperature Distribution Along Symmetry Plane
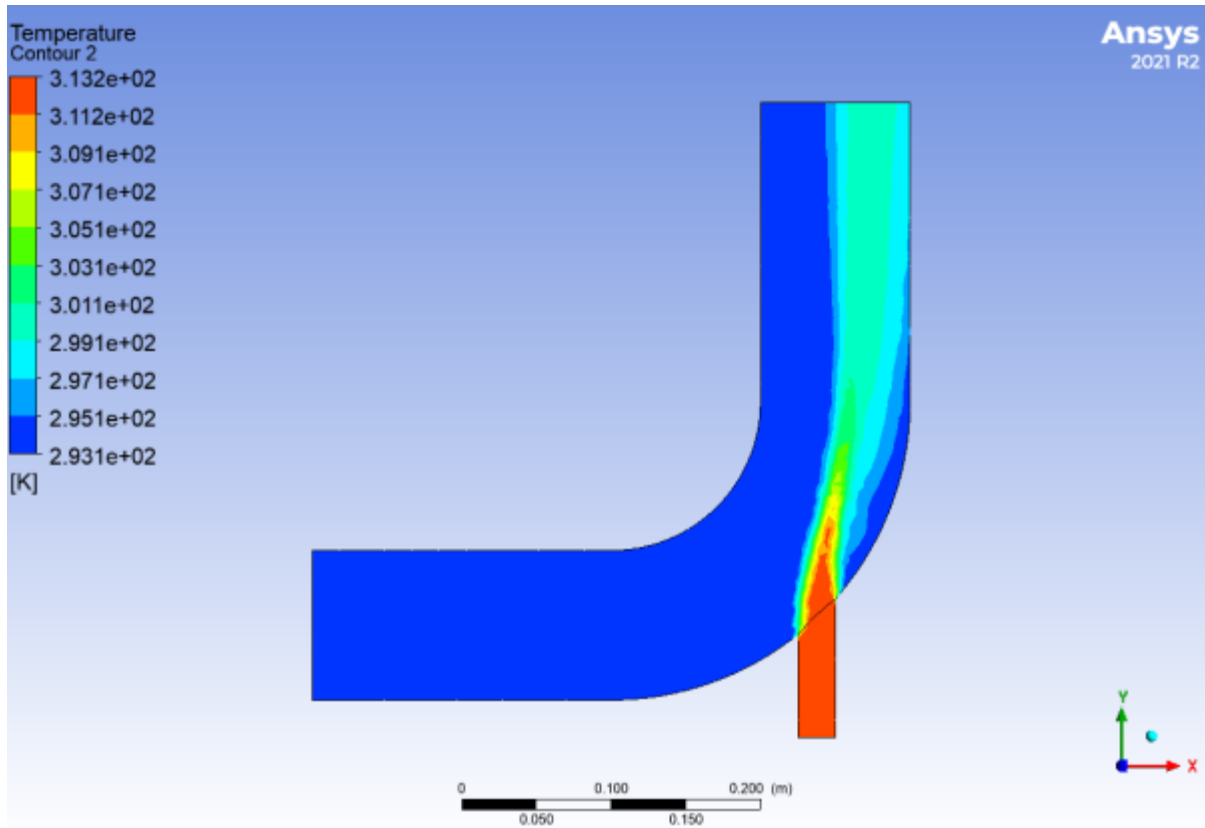
3. Close the CFD-Post application by selecting **File** → **Close CFD-Post** or by clicking the 'X' in the top right corner of the window.

Important:

Note that the CFD-Post state files are automatically saved when you exit CFD-Post and return to Ansys Workbench.

---

4. Save the `elbow-workbench` project in Ansys Workbench.

5. View the list of files generated by Ansys Workbench.

View → **Files**

Note the addition of the CFD-Post state file (`elbow.cst`) to the list of files. For more information about CFD-Post (and the files associated with it), see the CFD-Post documentation.

1.4.8. Duplicating the Fluent-Based Fluid Flow Analysis System

At this point, you have a completely defined fluid flow system that has a geometry, a computational mesh, a CFD setup and solution, and corresponding results. In order to study the effects upon the flow field that may occur if you were to alter the geometry, another fluid flow analysis is required. One approach would be to use the current system and change the geometry, however you would

overwrite the data from your previous simulation. A more suitable and effective approach would be to create a copy, or duplicate, of the current system, and then make the appropriate changes to the duplicate system.

In this step, you will create a duplicate of the original Fluent-based fluid flow system, then review the list of files generated by Ansys Workbench.

1. In the **Project Schematic**, right-click the title cell of the **Fluid Flow (Fluent)** system and select **Duplicate** from the context menu.

   Figure 1.21: Duplicating the Fluid Flow System
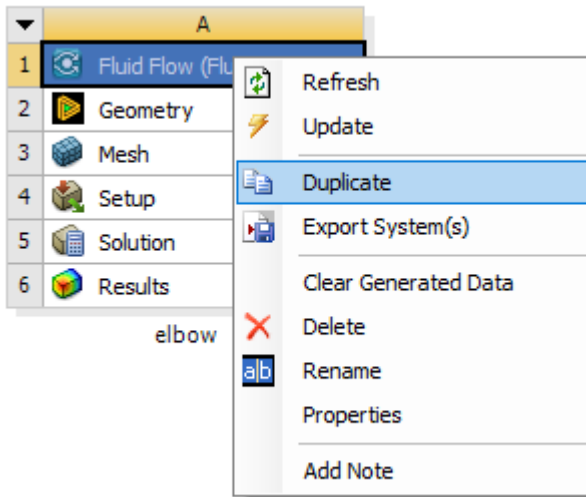
Figure 1.22: The Original Fluid Flow System and Its Duplicate

elbow

new-elbow
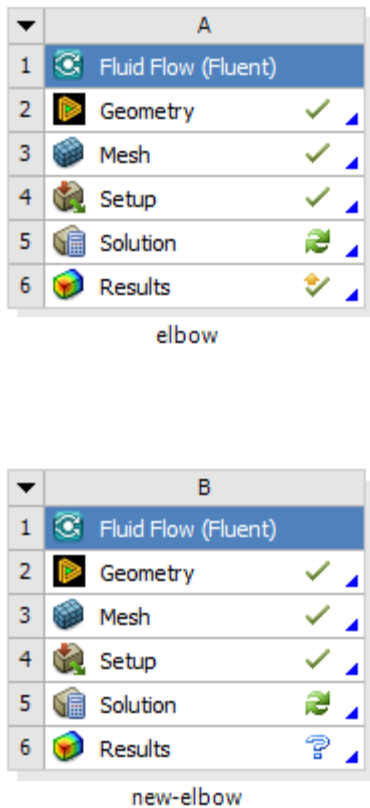
Note:

Notice that in the duplicated system, the state of the **Solution** cell indicates that the cell requires a refresh while the state of the **Results** cell indicates that the cell requires attention. This is because when a system is duplicated, the case and data files are not copied to the new system, therefore, the new system does not yet have solution data associated with it.

2. Rename the duplicated system to `new-elbow`.

3. Save the `elbow-workbench` project in Ansys Workbench.

1.4.9. Changing the Geometry in Ansys DesignModeler

Now that you have two separate, but equivalent, Fluent-based fluid flow systems to work from, you can make changes to the second system without impacting the original system. In this step, you will make a slight alteration to the elbow geometry in Ansys DesignModeler by changing the diameter of the smaller inlet, then review the list of files generated by Ansys Workbench.

1. Open Ansys DesignModeler.

Right-click the **Geometry** cell of the `new-elbow` system (cell `B2`) and select **Edit Geometry in DesignModeler...** from the context menu to display the geometry in Ansys DesignModeler.

2. Change the diameter of the small inlet (`velocity-inlet-small`).

   a. In the **Tree Outline**, select **Cylinder1** to open the **Details View** of the small inlet pipe.

   b. In the **Details View**, change the **FD10, Radius (>0)** value from `12.5` millimeters to `19` millimeters.

Details View		
Operation	Add Material	^
Origin Definition	Coordinates	
☐ FD3, Origin X Coordinate	137.5 mm	
☐ FD4, Origin Y Coordinate	-225 mm	
☐ FD5, Origin Z Coordinate	0 mm	
Axis Definition	Components	
☐ FD6, Axis X Component	0 mm	
☐ FD7, Axis Y Component	125 mm	
☐ FD8, Axis Z Component	0 mm	
☐ FD10, Radius (>0)	19 mm	
As Thin/Surface?	No	∨

   c. Click the **Generate** button to generate the geometry with your new values.

   Figure 1.23: Changing the Diameter of the Small Inlet in Ansys DesignModeler
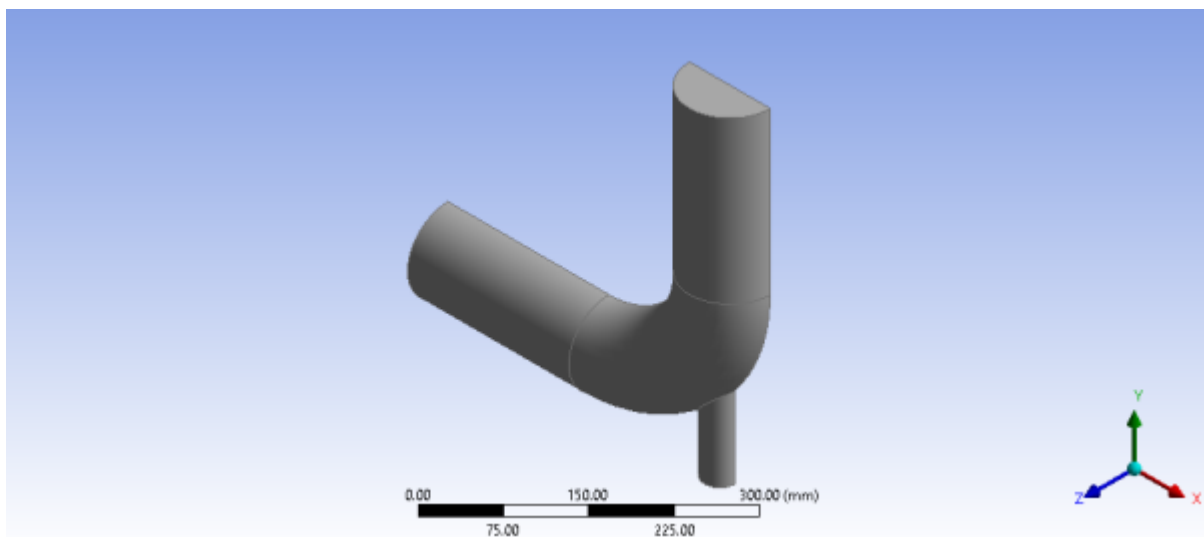
3. Close Ansys DesignModeler.

4. View the list of files generated by Ansys Workbench.

   View → Files

Note the addition of the geometry, mesh, and Ansys Fluent settings files now associated with the new, duplicated system.

1.4.10. Updating the Mesh in the Ansys Meshing Application

The modified geometry now requires a new computational mesh. The mesh settings for the duplicated system are retained in the duplicated system. In this step, you will update the mesh based on the mesh settings from the original system, then review the list of files generated by Ansys Workbench.

In the **Project Schematic**, right-click the **Mesh** cell of the `new-elbow` system (cell `B3`) and select **Update** from the context menu. This will update the mesh for the new geometry based on the mesh settings you specified earlier in the Ansys Meshing application without having to open the Ansys Meshing editor to regenerate the mesh.

It will take a few moments to update the mesh. Once the update is complete, the state of the **Mesh** cell is changed to up-to-date, symbolized by a green check mark.

For illustrative purposes of the tutorial, the new geometry and the new mesh are displayed below.

Figure 1.24: The Updated Geometry and Mesh in the Ansys Meshing Application
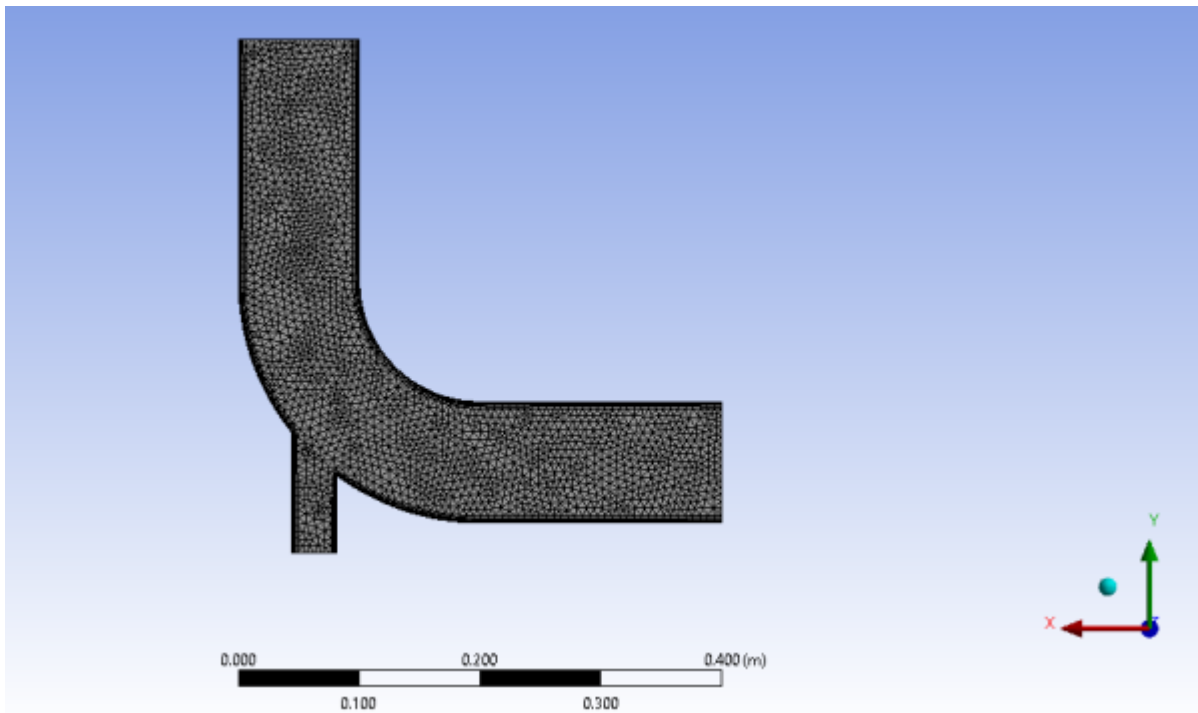
Inspecting the files generated by Ansys Workbench reveals the updated mesh file for the duplicated system.

View → Files

1.4.11. Calculating a New Solution in Ansys Fluent

Now that there is an updated computational mesh for the modified geometry in the duplicated system, a new solution must be generated using Ansys Fluent. In this step, you will revisit the settings within

Ansys Fluent, calculate another solution, view the new results, then review the list of files generated by Ansys Workbench.

1. Open Ansys Fluent.

    In the **Project Schematic**, right-click the **Setup** cell of the `new-elbow` system (cell B4) and select **Edit...** from the context menu. Since the mesh has been changed, you are prompted as to whether you want to load the new mesh into Ansys Fluent or not.
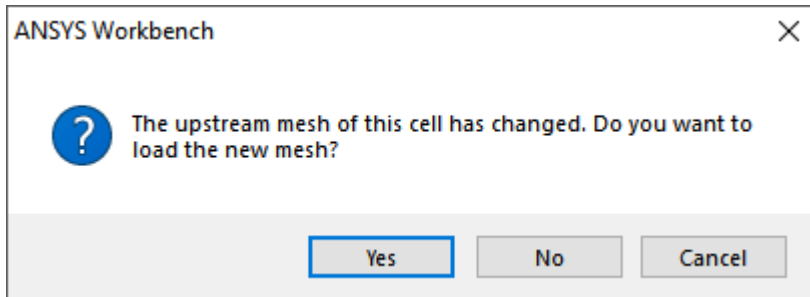
    Click **Yes** to continue, and click **Start** when Fluent Launcher is displayed in order to open Ansys Fluent.

2. Ensure that the unit of length is set to millimeters.

    Domain → Mesh → Units...

3. Check the mesh (optional).

    Setting Up Domain → Mesh → Check → Perform Mesh Check

4. Revisit the boundary conditions for the small inlet.

    Setup → Boundary Conditions → velocity-inlet-small Edit...

    In the **Velocity Inlet** dialog box for **velocity-inlet-small**, you must set the hydraulic diameter to 38 mm based on the new dimensions of the small inlet.

5. Re-initialize the solution.

    Solution → Initialization

    Keep the **Method** at the default of **Hybrid** and click **Initialize**.

6. Recalculate the solution.

    Solution → Run Calculation → Calculate

    Set the **No. of Iterations** to 250 and click **Calculate**.

7. Close Ansys Fluent.

8. Revisit the results of the calculations in CFD-Post.

Double-click the **Results** cell of the `new-elbow` fluid flow system to re-open CFD-Post where you can review the results of the new solution.

9. Close CFD-Post.

10. Save the `elbow-workbench` project in Ansys Workbench.

11. View the list of files generated by Ansys Workbench.

   View → Files

   Note the addition of the solution and state files now associated with new duplicated system.

1.4.12. Comparing the Results of Both Systems in CFD-Post

In this step, you will create a new **Results** system in Ansys Workbench, use that system to compare the solutions from each of the two Fluent-based fluid flow analysis systems in CFD-Post at the same time, then review the list of files generated by Ansys Workbench.

1. Create a **Results** system.

   In Ansys Workbench, drag a **Results** system from the **Component Systems** section of the **Toolbox** and drop it into the **Project Schematic**, next to the fluid flow systems.

   Figure 1.25: The New Results System in the Project Schematic
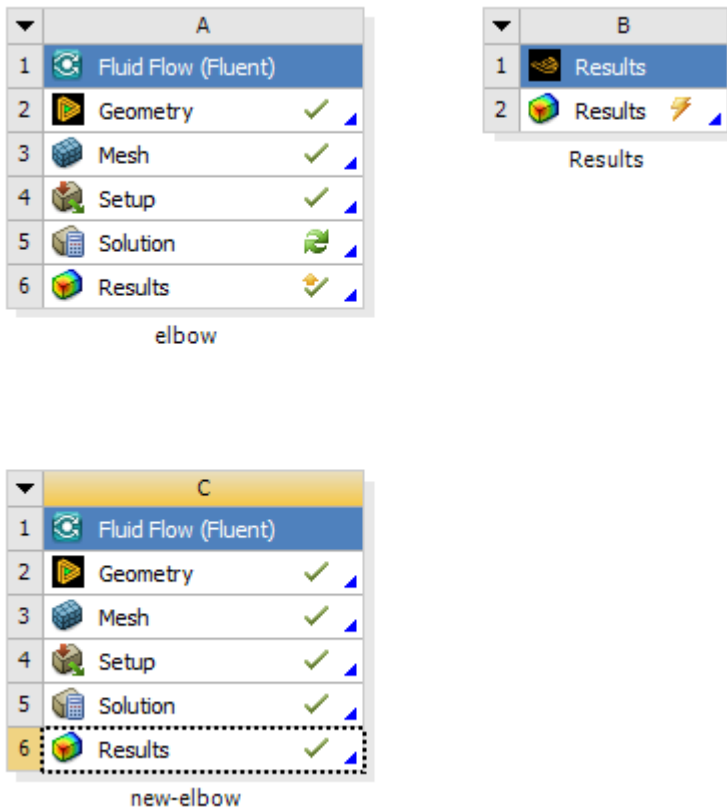
2. Add the solutions of each of the systems to the **Results** system.

    a. Select the **Solution** cell in the first Fluid Flow analysis system (cell `A5`) and drag it over the **Results** cell in the **Results** system (cell `C2`). This creates a transfer data connection between the two systems.

Figure 1.26: Connecting the First Fluid Flow System to the New Results System
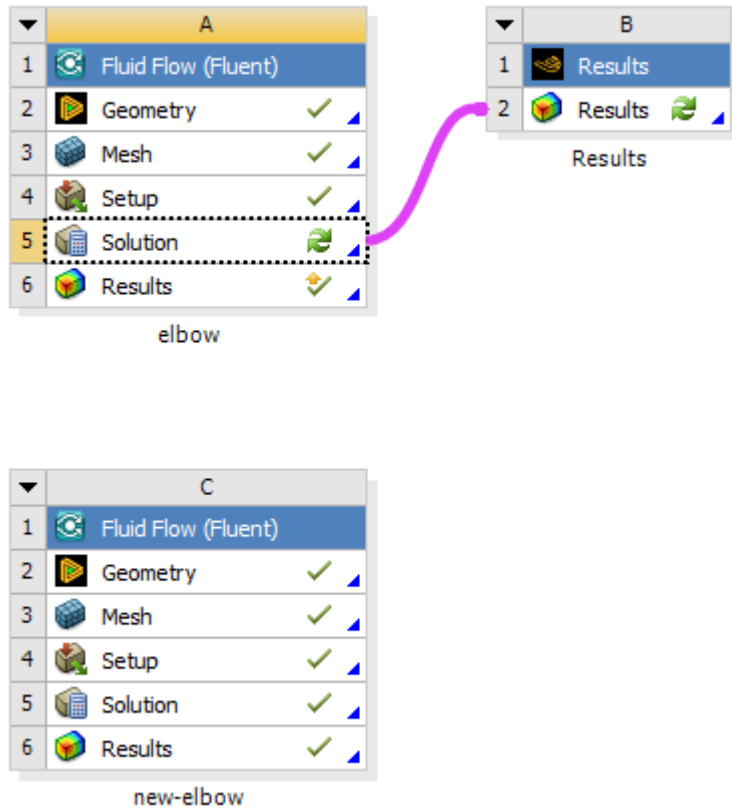
    b. Select the **Solution** cell in the second Fluid Flow analysis system (cell `B5`) and drag it over the **Results** cell in the **Results** system (cell `C2`). This creates a transfer data connection between the two systems.

Figure 1.27: Connecting the Second Fluid Flow System to the New Results System
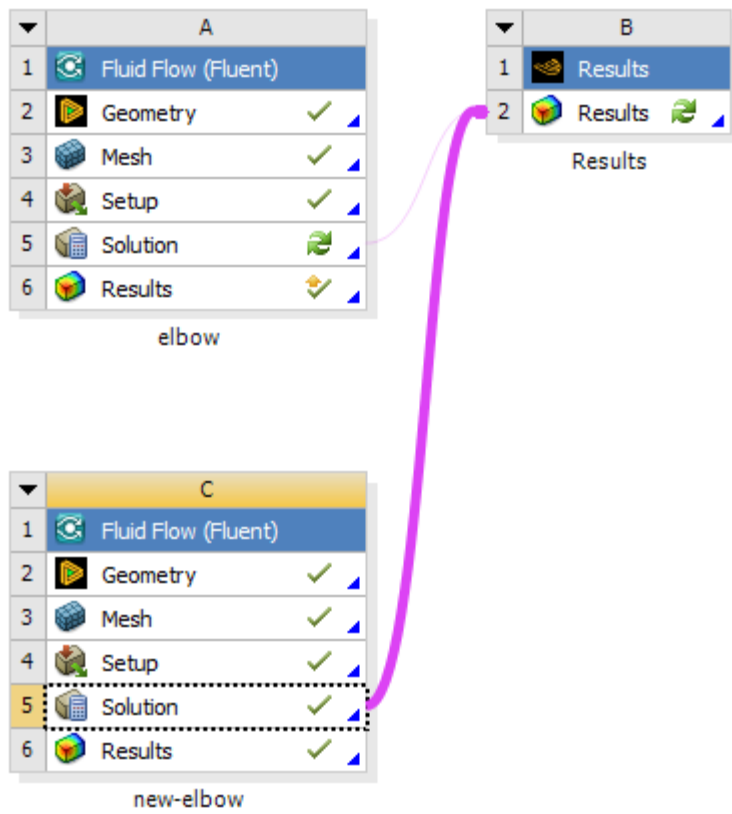
3. Open CFD-Post to compare the results of the two fluid flow systems.

   Now that the two fluid flow systems are connected to the **Results** system, double-click the **Results** cell in the **Results** system (cell C2) to open CFD-Post. Within CFD-Post, both geometries are displayed side by side.

62

Figure 1.28: CFD-Post with Both Fluid Flow Systems Displayed
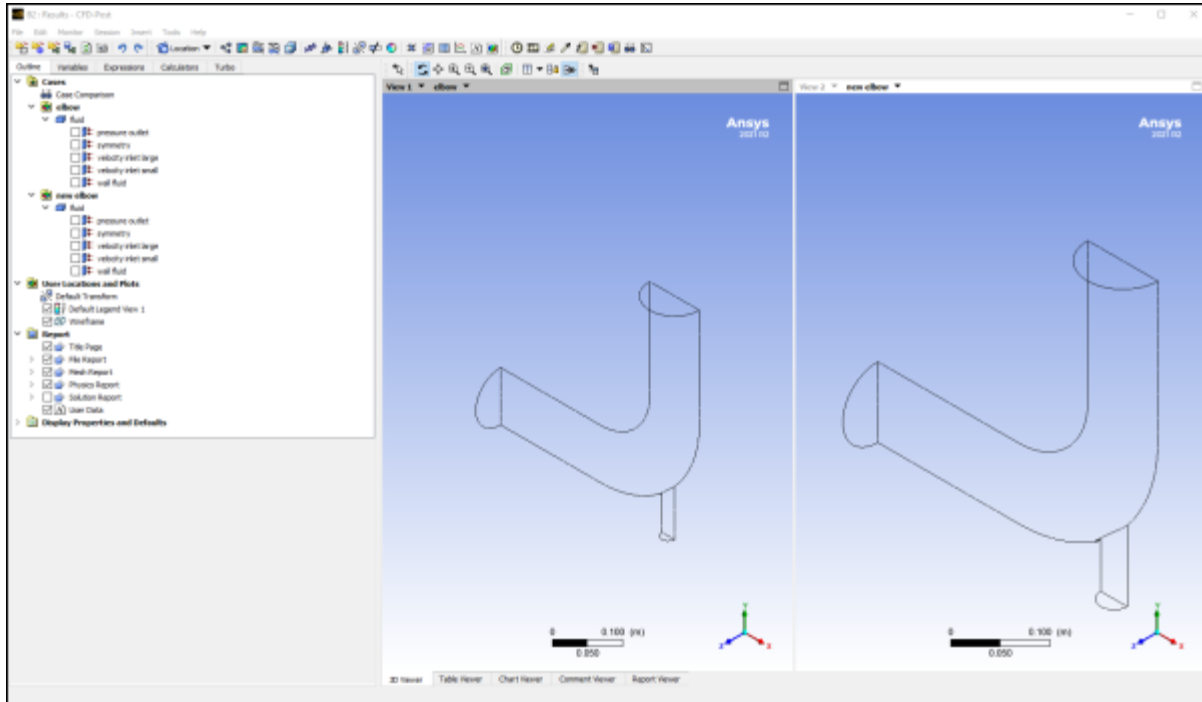
a.   Re-orient the display.

In each view, click the blue Z axis on the axis triad in the bottom right hand corner of the graphics display to orient the display so that the view is of the front of the elbow geometry.

> **Important:**
>
> Alternatively, you can select the synchronization tool ( ) in the **3D Viewer Toolbar** to synchronize the views, so that when you re-orient one view, the other view is automatically updated.

b.   Display contours of velocity magnitude on the symmetry plane.

   i.   Insert a contour object.

   Insert → Contour

   This displays the **Insert Contour** dialog box.

   ii.   Keep the default name of the contour (Contour 1) and click **OK** to close the dialog box. This displays the **Details of Contour 1** view below the **Outline** view in CFD-Post. This view contains all of the settings for a contour object.

   iii.   In the **Geometry** tab, from the **Domains** list, select **fluid**.

   iv.   In the **Locations** list, select **symmetry**.

   v.   In the **Variable** list, select **Velocity**.

vi. Click **Apply**. The velocity contours are displayed in each view.

Note:

To better visualize the velocity display, you can clear the **Wireframe** view option
under **User Locations and Plots** in the **Outline** tree view.

Figure 1.29: CFD-Post Displaying Velocity Contours for Both Geometries
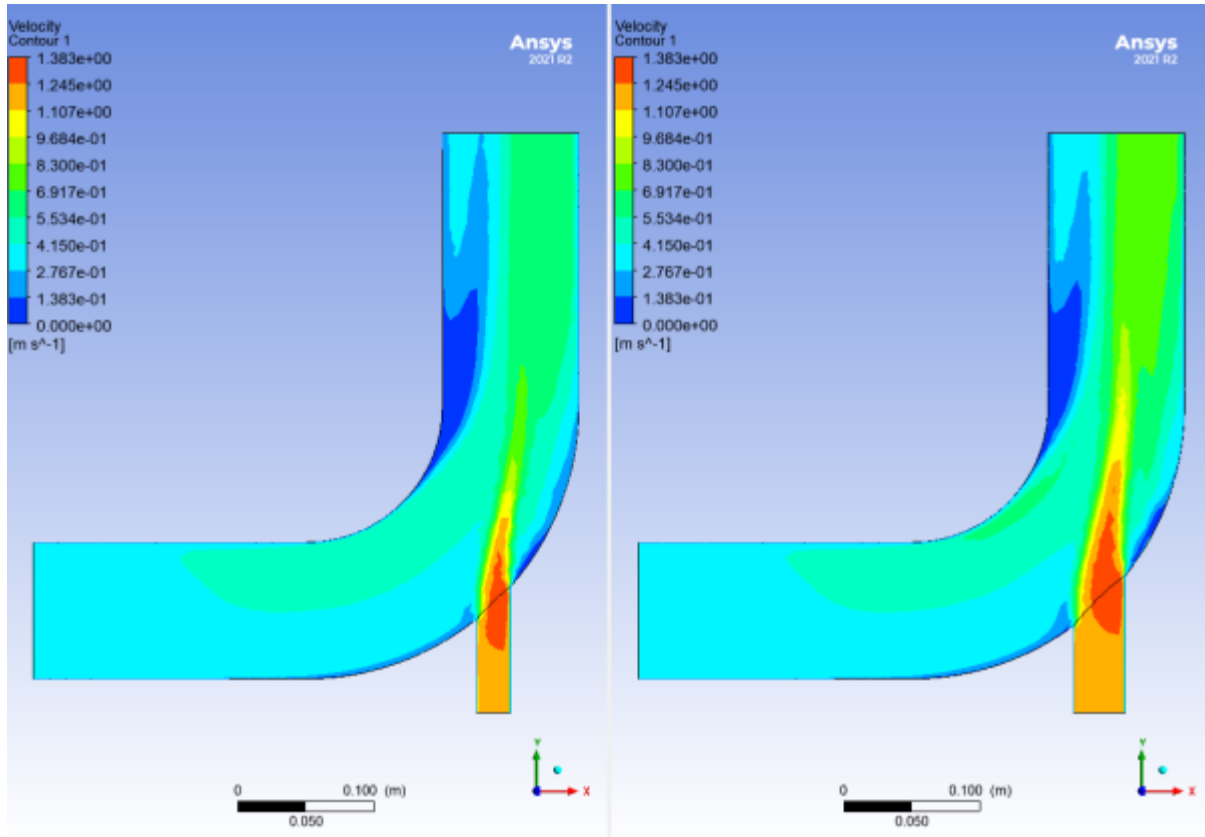
c. Display contours of temperature on the symmetry plane.

i. In the **Outline** tree view, under **User Locations and Plots**, deselect the **Contour 1** object
to hide the first contour display in CFD-Post.

ii. Insert another contour object.

Insert → Contour

This displays the **Insert Contour** dialog box.

iii. Keep the default name of the contour (Contour 2) and click **OK** to close the dialog box.
This displays the **Details of Contour 2** view below the **Outline** view in CFD-Post.

iv. In the **Geometry** tab, select **fluid** in the **Domains** list.

v. Select **symmetry** in the **Locations** list.

vi. Select **Temperature** in the **Variable** list.

vii. Click **Apply**. The temperature contours are displayed in each view.

Figure 1.30: CFD-Post Displaying Temperature Contours for Both Geometries
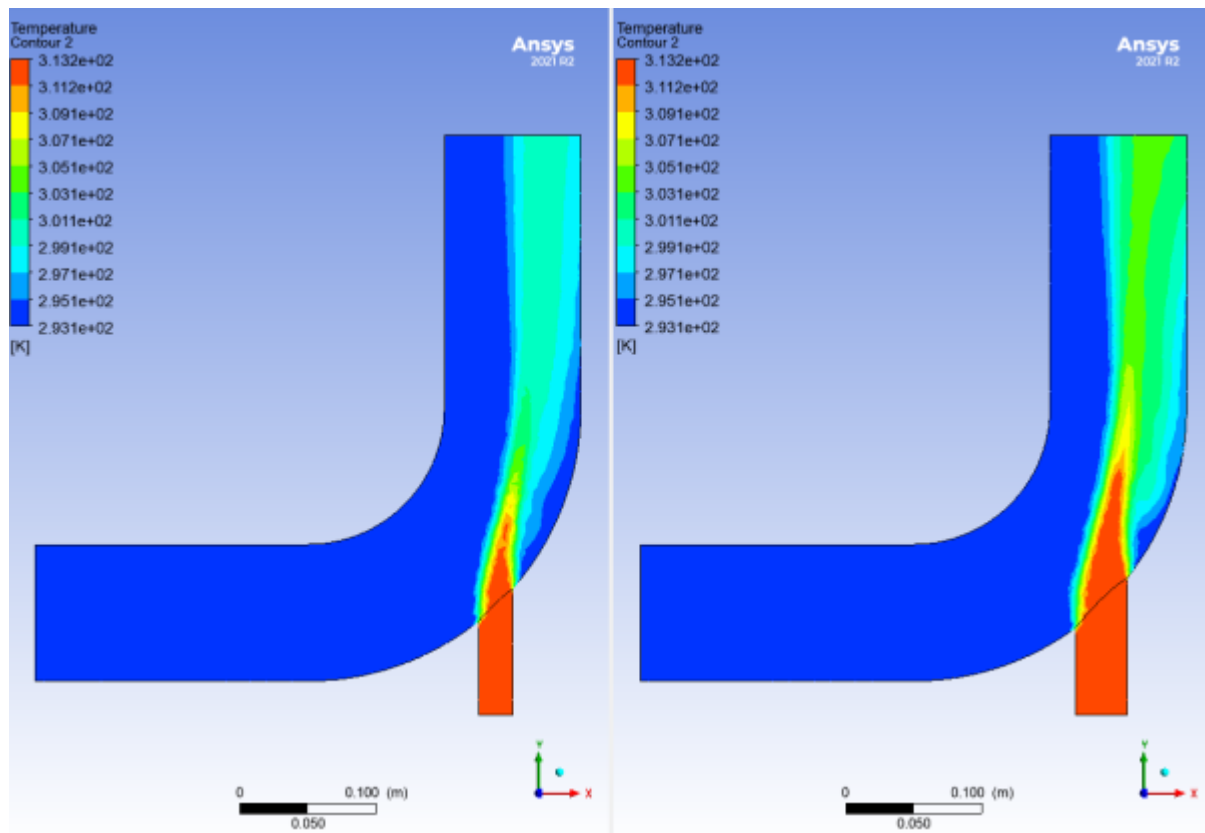
4. Close the CFD-Post application.

5. Save the `elbow-workbench` project in Ansys Workbench.

6. View the list of files associated with your project using the **Files** view.

   View → Files

   Note the addition of the **Results** system and its corresponding files.

1.5. Summary

In this tutorial, portions of Ansys Workbench were used to compare the fluid flow through two slightly different geometries. Ansys DesignModeler was used to create a mixing elbow geometry, Ansys Meshing was used to create a computational mesh, Ansys Fluent was used to calculate the fluid flow throughout the geometry using the computational mesh, and CFD-Post was used to analyze the results. In addition, the geometry was altered, a new mesh was generated, and a new solution was calculated. Finally, Ansys Workbench was set up so that CFD-Post could directly compare the results of both calculations at the same time.

Chapter 2: Parametric Analysis in Ansys Workbench Using Ansys Fluent

This tutorial is divided into the following sections:

2.1. Introduction

This tutorial illustrates using an Ansys Fluent fluid flow system in Ansys Workbench to set up and solve a three-dimensional turbulent fluid flow and heat transfer problem in an automotive heating, ventilation, and air conditioning (HVAC) duct system. Ansys Workbench uses parameters and design points to allow you to run optimization and what-if scenarios. You can define both input and output parameters in Ansys Fluent that can be used in your Ansys Workbench project. You can also define parameters in other applications including Ansys DesignModeler and Ansys CFD-Post. Once you have defined parameters for your system, a **Parameters** cell is added to the system and the **Parameter Set** bus bar is added to your project. This tutorial is designed to introduce you to the parametric analysis utility available in Ansys Workbench.

The tutorial starts with a Fluid Flow (Fluent) analysis system with pre-defined geometry and mesh components. Within this tutorial, you will redefine the geometry parameters created in Ansys Design-Modeler by adding constraints to the input parameters. You will use Ansys Fluent to set up and solve the CFD problem. While defining the problem set-up, you will also learn to define input parameters in Ansys Fluent. The tutorial will also provide information on how to create output parameters in Ansys CFD-Post.

This tutorial demonstrates how to do the following:

- Add constraints to the Ansys DesignModeler input parameters.

- Create an Ansys Fluent fluid flow analysis system in Ansys Workbench.

- Set up the CFD simulation in Ansys Fluent, which includes:

  - Setting material properties and boundary conditions for a turbulent forced convection problem.

  - Defining input parameters in Fluent

- Define output parameters in CFD-Post

- Create additional design points in Ansys Workbench.

- Run multiple CFD simulations by updating the design points.

- Analyze the results of each design point project in Ansys CFD-Post and Ansys Workbench.

Important:

The mesh and solution settings for this tutorial are designed to demonstrate a basic parameterization simulation within a reasonable solution time-frame. Ordinarily, you would use additional mesh and solution settings to obtain a more accurate solution.

2.2. Prerequisites

This tutorial assumes that you are already familiar with the Ansys Workbench interface and its project workflow (for example, Ansys DesignModeler, Ansys Meshing, Ansys Fluent, and Ansys CFD-Post). This tutorial also assumes that you have completed the introductory Ansys Fluent tutorials, and that you are familiar with the Ansys Fluent graphical user interface. Some steps in the setup and solution procedure will not be shown explicitly.

2.3. Problem Description

In the past, evaluation of vehicle air conditioning systems was performed using prototypes and testing their performance in test labs. However, the design process of modern vehicle air conditioning (AC) systems improved with the introduction of Computer Aided Design (CAD), Computer Aided Engineering (CAE) and Computer Aided Manufacturing (CAM). The AC system specification will include minimum performance requirements, temperatures, control zones, flow rates, and so on. Performance testing using CFD may include fluid velocity (air flow), pressure values, and temperature distribution. Using CFD enables the analysis of fluid through very complex geometry and boundary conditions.

As part of the analysis, a designer can change the geometry of the system or the boundary conditions such as the inlet velocity, flow rate, and so on, and view the effect on fluid flow patterns. This tutorial illustrates the AC design process on a representative automotive HVAC system consisting of both an evaporator for cooling and a heat exchanger for heating requirements. This HVAC system is symmetric, so the geometry has been simplified using a plane of symmetry to reduce computation time.

Figure 2.1: Automotive HVAC System
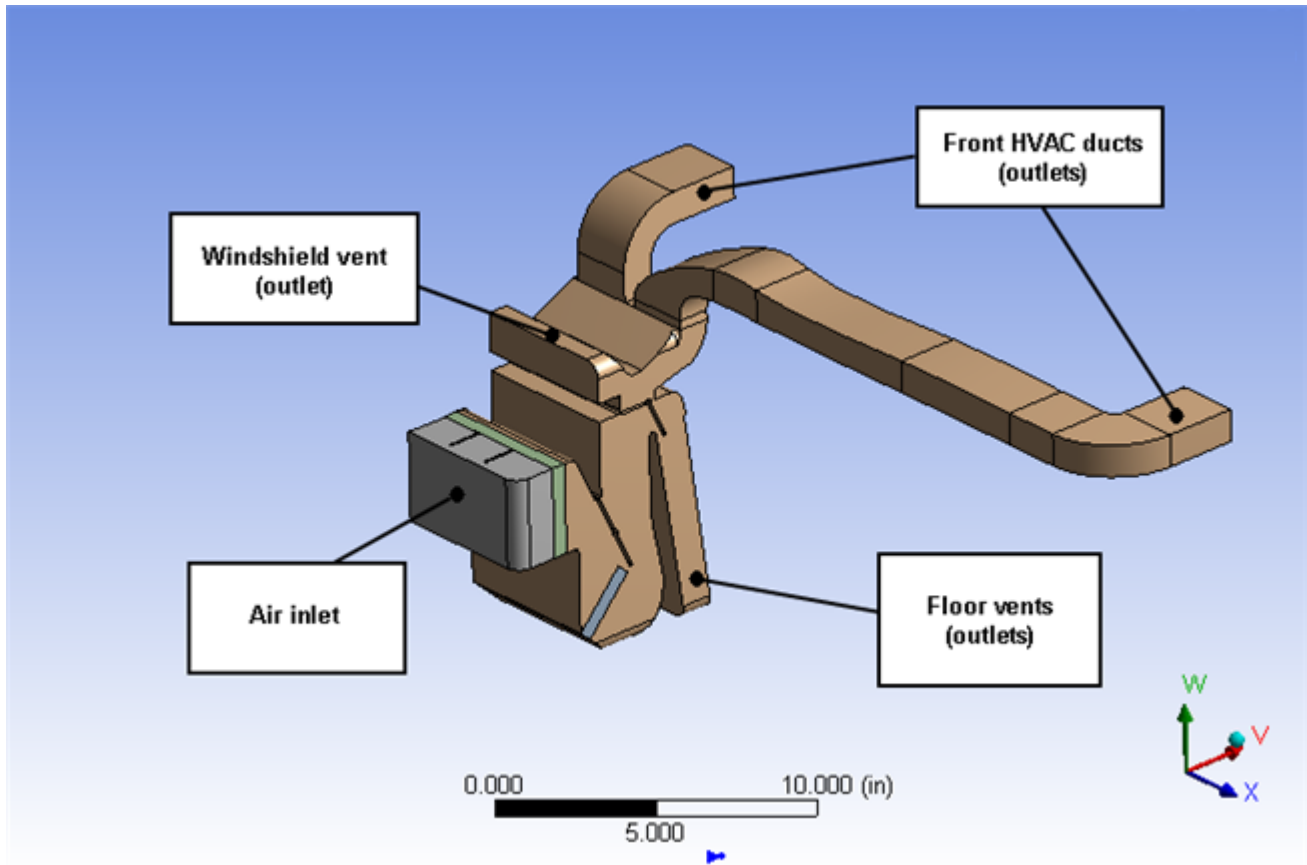
Figure 2.2: HVAC System Valve Location Details
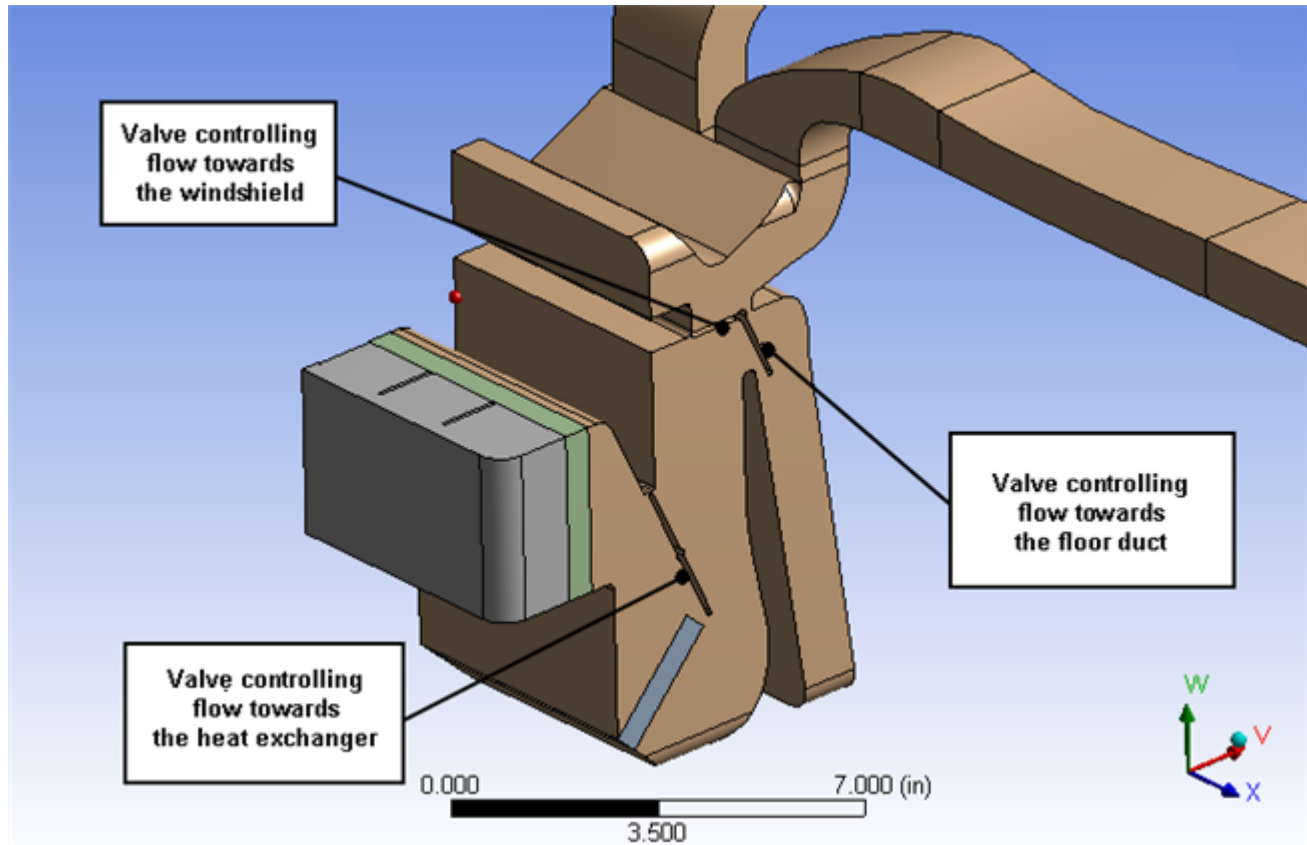
Figure 2.1: Automotive HVAC System (p. 69) shows a representative automotive HVAC system. The system has three valves (as shown in Figure 2.2: HVAC System Valve Location Details (p. 70)), which control the flow in the HVAC system. The three valves control:

- Flow over the heat exchanger coils

- Flow towards the duct controlling the flow through the floor vents

- Flow towards the front vents or towards the windshield

Air enters the HVAC system at 310 K with a velocity of 0.5 m/sec through the air inlet and passes to the evaporator and then, depending on the position of the valve controlling flow to the heat exchanger, flows over or bypasses the heat exchanger. Depending on the cooling and heating requirements, either the evaporator or the heat exchanger would be operational, but not both at the same time. The position of the other two valves controls the flow towards the front panel, the windshield, or towards the floor ducts.

The motion of the valves is constrained. The valve controlling flow over the heat exchanger varies between 25° and 90°. The valve controlling the floor flow varies between 20° and 60°. The valve controlling flow towards front panel or windshield varies between 15° and 175°.

The evaporator load is about 200 W in the cooling cycle. The heat exchanger load is about 150 W.

This tutorial illustrates the easiest way to analyze the effects of the above parameters on the flow pattern/distribution and the outlet temperature of air (entering the passenger cabin). Using the parametric

analysis capability in Ansys Workbench, a designer can check the performance of the system at various design points.

Figure 2.3: Flow Pattern for the Cooling Cycle
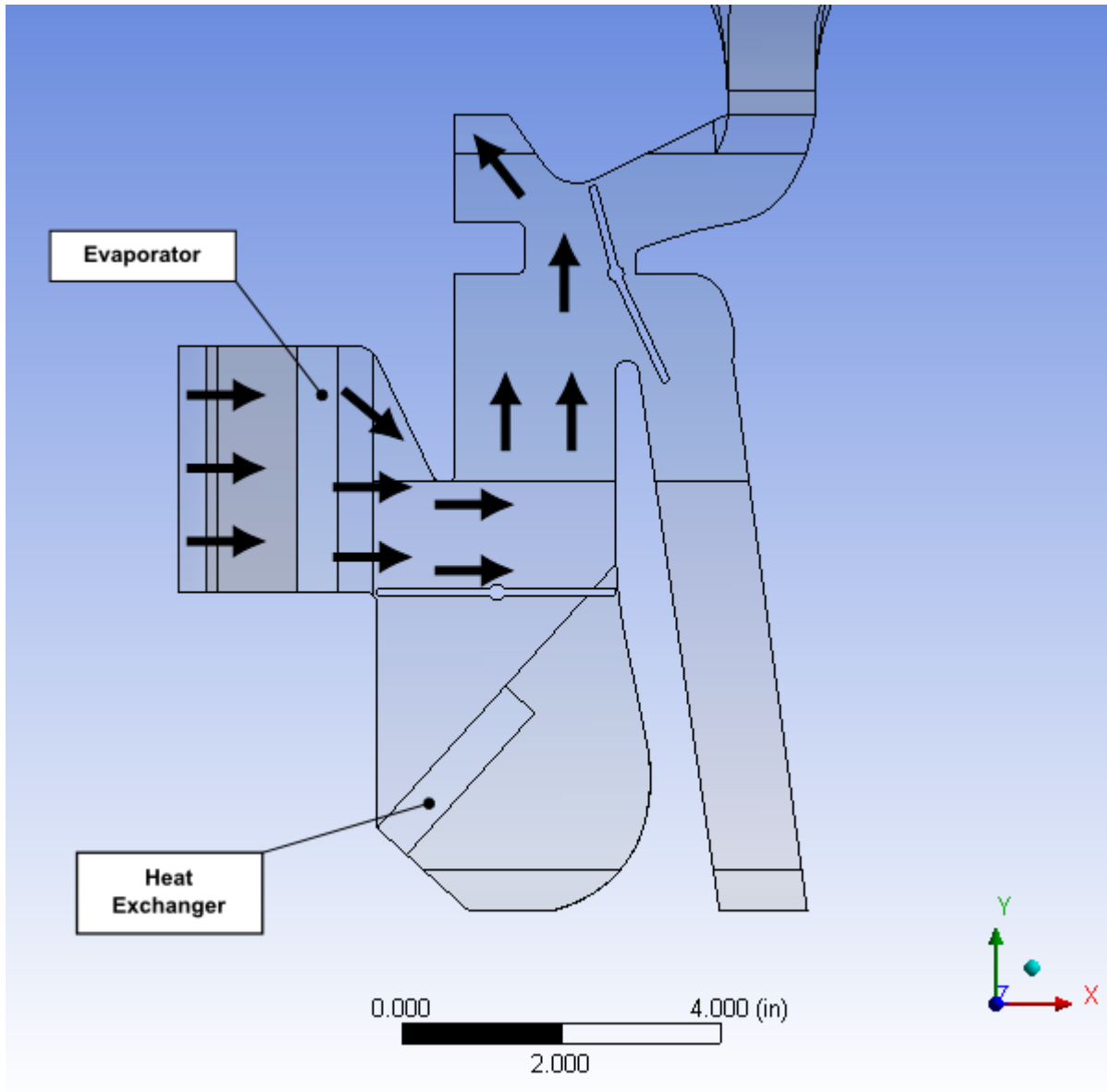
2.4. Setup and Solution

The following sections describe the setup and solution steps for this tutorial:

2.4.1. Preparation

To prepare for running this tutorial:

1. Download the `workbench_parameter.zip` file here.

2. Unzip the `workbench_parameter.zip` file to your working folder.

   The extracted `workbench_parameter` folder contains a single archive file `fluent-workbench-param.wbpz` that includes all supporting input files of the starting Ansys Workbench project and a folder called `final_project_files` that includes the archived final version of the project. The final result files incorporate Ansys Fluent and Ansys CFD-Post settings and all already defined design points (all that is required is to update the design points in the project to generate corresponding solutions).

2.4.2. Adding Constraints to Ansys DesignModeler Parameters in Ansys Workbench

In this step, you will start Ansys Workbench, open the project file, review existing parameters, create new parameters, and add constraints to existing Ansys DesignModeler parameters.

1. From the Windows **Start** menu, select **Start > ANSYS 2021 R2 > Workbench 2021 R2** to start Ansys Workbench.

   This displays the Ansys Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox**, and the components of the analysis system are displayed in the **Project Schematic**.

   > **Note:**
   >
   > When you first start Ansys Workbench, the **Getting Started** message window is displayed, offering assistance through the online help for using the application. You can keep the window open, or close it by clicking **OK**. If you need to access the online help at any time, use the **Help** menu, or press the **F1** key.

2. Restore the archive of the starting Ansys Workbench project to your working directory.

   File → Open...

   Browse to your working directory, select the project archive file `fluent-workbench-param.wbpz`, and click **Open**.

   Now that the project archive has been restored, the project will automatically open in Ansys Workbench.

Figure 2.4: The Project Loaded into Ansys Workbench
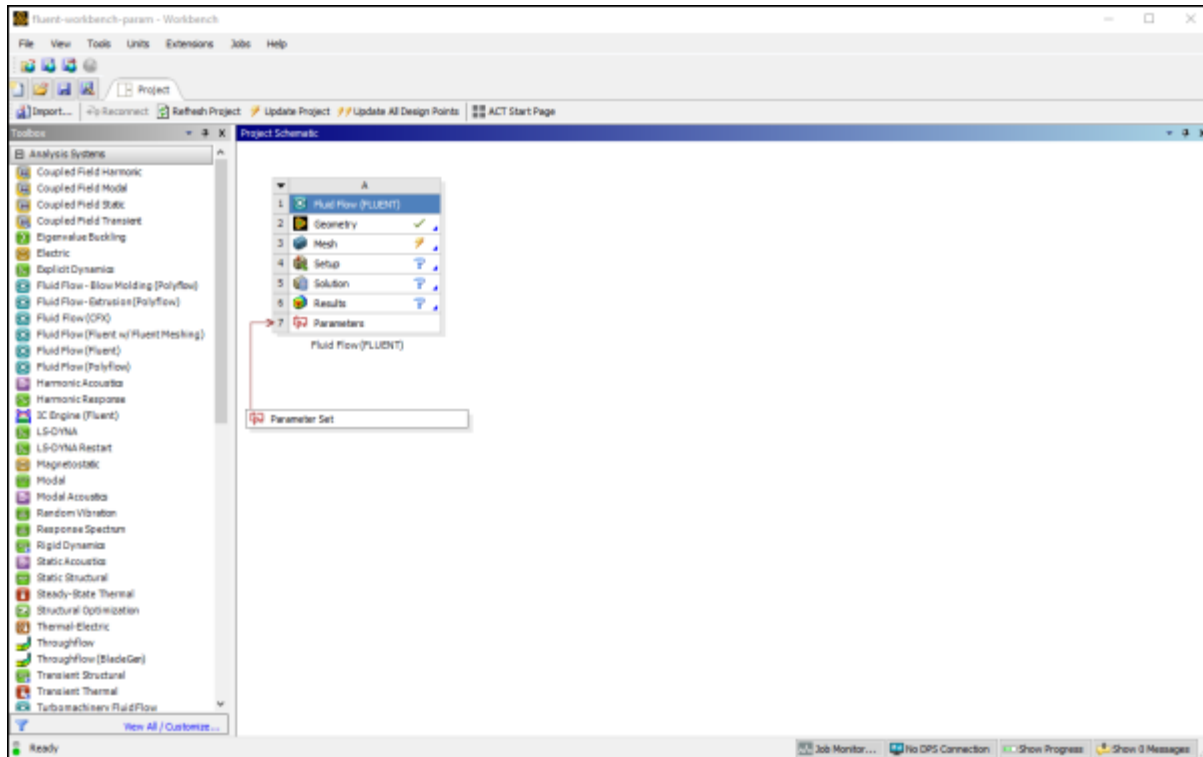
The project (`fluent-workbench-param.wbpj`) already has a Fluent-based fluid flow analysis system that includes the geometry and mesh, as well as some predefined parameters. You will first examine and edit parameters within Workbench, then later proceed to define the fluid flow model in Ansys Fluent.

3. Review the input parameters that have already been defined in Ansys DesignModeler.

   a. Double-click the **Parameter Set** bus bar in the Ansys Workbench **Project Schematic** to open the **Parameters Set** tab.

   > **Note:**
   >
   > To return to viewing the **Project Schematic**, click the **Project** tab.

   b. In the **Outline of All Parameters** view (Figure 2.5: Parameters Defined in Ansys DesignModeler (p. 74)), review the following existing parameters:

   - The parameter `hcpos` represents the valve position that controls the flow over the heat exchanger. When the valve is at an angle of 25°, it allows the flow to pass over the heat exchanger. When the angle is 90°, it completely blocks the flow towards the heat exchanger. Any value in between allows some flow to pass over the heat exchanger giving a mixed flow condition.

   - The parameter `ftpos` represents the valve position that controls flow towards the floor duct. When the valve is at an angle of 20°, it blocks the flow towards the floor duct and when the valve angle is 60°, it unblocks the flow.

- The parameter `wsfpos` represents the valve position that controls flow towards the windshield and the front panel. When the valve is at an angle of 15°, it allows the entire flow to go towards the windshield. When the angle is 90°, it completely blocks the flow towards windshield as well as the front panel. When the angle is 175°, it allows the flow to go towards the windshield and the front panel.

Figure 2.5: Parameters Defined in Ansys DesignModeler

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Fluid Flow (FLUENT) (A1)			
4	P1	hcpos	90	
5	P2	ftpos	25	
6	P3	wsfpos	175	
*	New input parameter	New name	New expression	
8	Output Parameters			
*	New output parameter		New expression	
10	Charts			

4. In the **Outline of All Parameters** view, create three new named input parameters.

   a. In the row that contains **New input parameter**, click the parameter table cell with **New name** (under the **Parameter Name** column) and enter `input_hcpos`. Note the ID of the parameter that appears in column **A** of the table. For the new input parameter, the parameter **ID** is **P4**. In the **Value** column, enter **15**.

   b. In a similar manner, create two more parameters named `input_ftpos` and `input_wsfpos`. In the **Value** column, enter **25**, and **90** for each new parameter (**P5** and **P6**), respectively.

Figure 2.6: New Parameters Defined in Ansys Workbench

	A	B	C	D
	ID	**Parameter Name**	**Value**	**Unit**
1				
2	⊟ Input Parameters			
3	⊟ 🗘 Fluid Flow (FLUENT) (A1)			
4	P1	hcpos	90	
5	P2	ftpos	25	
6	P3	wsfpos	175	
7	P4	input_hcpos	15	
8	P5	input_ftpos	25	
9	P6	input_wspos	90	
*	New input parameter	New name	New expression	
11	⊟ Output Parameters			
*	New output parameter		New expression	
13	Charts			

5. Select the row (or any cell in the row) that corresponds to the `hcpos` parameter. In the **Properties of Outline** view, change the value of the `hcpos` parameter in the **Expression** field from 90 to the expression `min(max(25,P4),90)`. This puts a constraint on the value of `hcpos`, so that the value always remains between 25° and 90°. The redefined parameter `hcpos` is automatically passed to Ansys DesignModeler. Alternatively the same constraint can also be set using the expression `max(25, min(P4,90))`.

After defining this expression, the parameter becomes a derived parameter that is dependent on the value of the parameter `input_hcpos` with ID **P4**. The derived parameters are unavailable for editing in the **Outline of All Parameters** view and could be redefined only in the **Properties of Outline** view.

Important:

When entering expressions, you must use the list and decimal delimiters associated with your selected language in the Workbench regional and language settings, which correspond to the regional settings on your machine. The instructions in this tutorial assume that your systems uses "." as a decimal separator and "," as a list separator.

Figure 2.7: Constrained Parameter `hcpos`

	A	B	C	D
	Outline of All Parameters			
1	ID	Parameter Name	Value	Unit
2	⊟ Input Parameters			
3	⊟ ⓒ Fluid Flow (FLUENT) (A1)			
4	p̂ P1	hcpos	25	
5	p̂ P2	ftpos	25	
6	p̂ P3	wsfpos	175	
7	p̂ P4	input_hcpos	15	
8	p̂ P5	input_ftpos	25	
9	p̂ P6	input_wspos	90	
*	p̂ New input parameter	New name	New expression	
11	⊟ Output Parameters			
*	p̄ New output parameter		New expression	
13	Charts			

	A	B
	Properties of Outline B4: P1	
1	Property	Value
2	⊟ General	
3	Expression	min(max(25,P4),90)
4	Usage	Input
5	Description	
6	Error Message	
7	Expression Type	Derived
8	Quantity Name	Dimensionless

6. Select the row or any cell in the row that corresponds to the `ftpos` parameter and create a similar expression for `ftpos`: `min(max(20,P5),60)`.

Figure 2.8: Constrained Parameter `ftpos`

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	☐ Input Parameters			
3	☐ ⊙ Fluid Flow (FLUENT) (A1)			
4	ⓟ P1	hcpos	25	
5	ⓟ P2	ftpos	25	
6	ⓟ P3	wsfpos	175	
7	ⓟ P4	input_hcpos	15	
8	ⓟ P5	input_ftpos	25	
9	ⓟ P6	input_wspos	90	
*	ⓟ New input parameter	New name	New expression	
11	☐ Output Parameters			
*	ⓟ New output parameter		New expression	
13	Charts			

Properties of Outline B5: P2

	A	B
1	Property	Value
2	☐ General	
3	Expression	min(max(20,P5),60)
4	Usage	Input
5	Description	
6	Error Message	
7	Expression Type	Derived
8	Quantity Name	Dimensionless

7. Create a similar expression for `wsfpos`: `min(max(15,P6),175)`.

Figure 2.9: Constrained Parameter `wsfpos`

	A	B	C	D
		ID	Parameter Name	Value	Unit
1		ID	Parameter Name	Value	Unit
2	⊟ Input Parameters				
3	⊟ 🄲 Fluid Flow (FLUENT) (A1)				
4	🄿 P1	hcpos	25	
5	🄿 P2	ftpos	25	
6	🄿 P3	wsfpos	90	
7	🄿 P4	input_hcpos	15	
8	🄿 P5	input_ftpos	25	
9	🄿 P6	input_wspos	90	
*	🄿 New input parameter	New name	New expression	
11	⊟ Output Parameters			
*	🄿 New output parameter		New expression	
13	Charts			

Properties of Outline B6: P3

	A	B
1	Property	Value
2	⊟ General	
3	Expression	min(max(15,P6),175)
4	Usage	Input
5	Description	
6	Error Message	
7	Expression Type	Derived
8	Quantity Name	Dimensionless

8. Click the **X** on the right side of the **Parameters Set** tab to close it and return to the **Project Schematic**.

   Note the new status of the cells in the Fluid Flow (Fluent) analysis system. Since we have changed the values of `hcpos`, `ftpos`, and `wsfpos` to their new expressions, the **Geometry** and **Mesh** cells now indicates Refresh Required (🔄).

9. Update the **Geometry** and **Mesh** cells.

   a. Right-click the **Geometry** cell and select the **Update** option from the context menu.

b. Likewise, right-click the **Mesh** cell and select the **Refresh** option from the context menu. Once the cell is refreshed, then right-click the **Mesh** cell again and select the **Update** option from the context menu.

10. Save the project in Ansys Workbench.

In the main menu, select **File** → **Save**

2.4.3. Setting Up the CFD Simulation in Ansys Fluent

Now that you have edited the parameters for the project, you will set up a CFD analysis using Ansys Fluent. In this step, you will start Ansys Fluent, and begin setting up the CFD simulation.

2.4.3.1. Starting Ansys Fluent

In the Ansys Workbench **Project Schematic**, double-click the **Setup** cell in the Ansys Fluent fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

When Ansys Fluent is first started, Fluent Launcher is displayed, allowing you to view and/or set certain Ansys Fluent start-up options.

Fluent Launcher allows you to decide which version of Ansys Fluent you will use, based on your geometry and on your processing capabilities.

Figure 2.10: Ansys Fluent Launcher
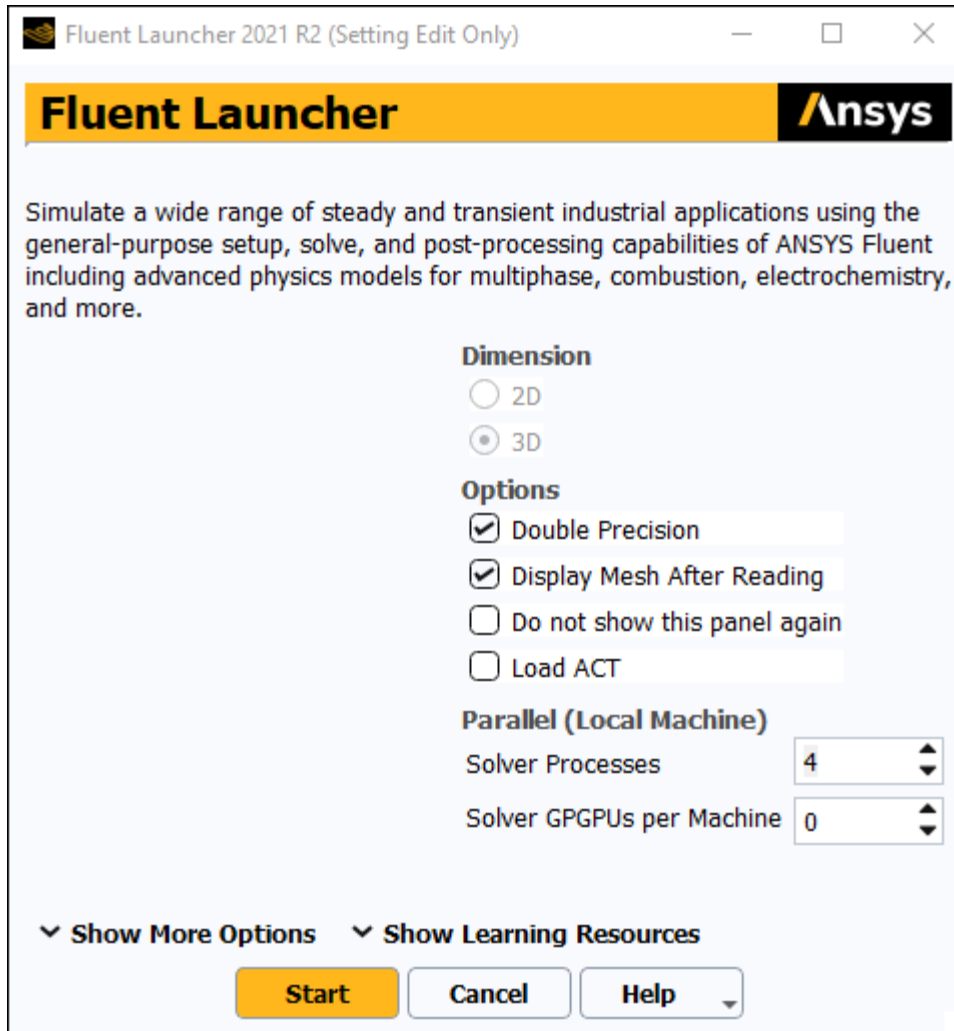
1. Ensure that the proper options are enabled.

Important:

Note that the **Dimension** setting is already filled in and cannot be changed, since Ansys Fluent automatically sets it based on the mesh or geometry for the current system.

---

   a. Ensure that the **Double Precision** option is enabled.

   b. Set **Solver Processes** to 4 under **Parallel (Local Machine)**.

Note:

Fluent will retain your preferences for future sessions.

---

2. Click **Start** to launch Ansys Fluent.

Figure 2.11: The Ansys Fluent Application
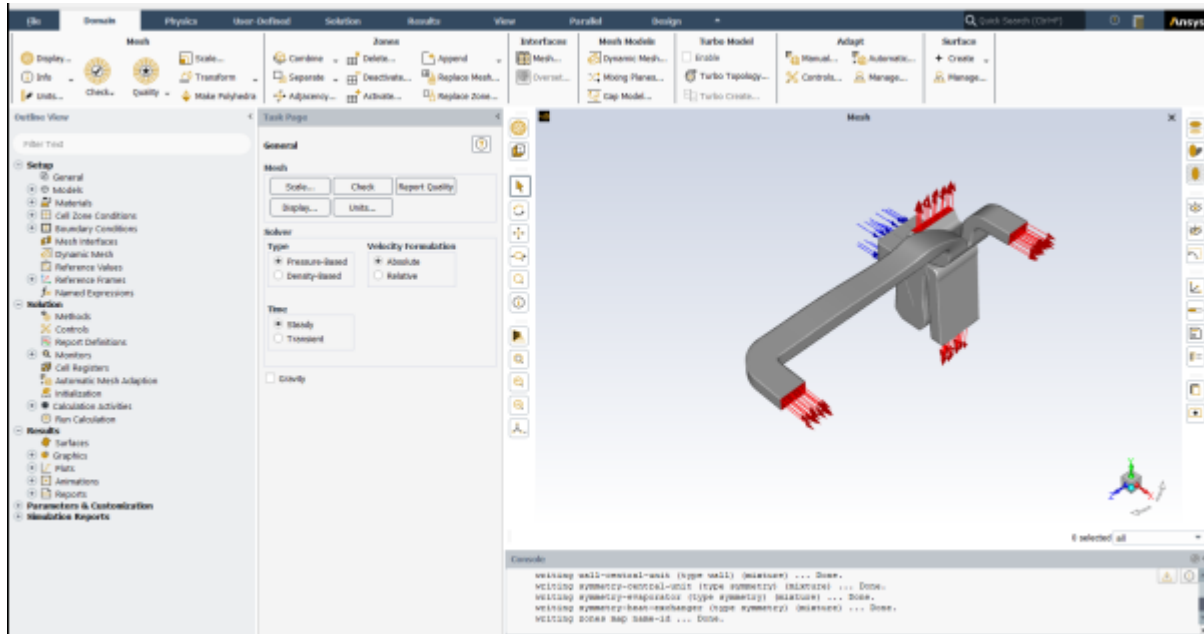
2.4.3.2. Setting Up Physics

1. In the **Solver** group of the **Physics** ribbon tab, retain the default selection of the steady pressure-based solver.

   ▸▸◂◂ **Physics → Solver**
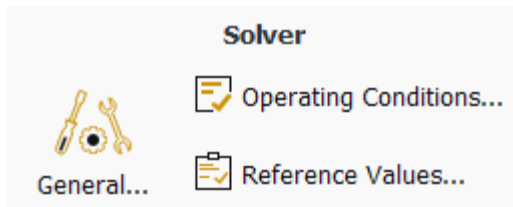
2. Set up your models for the CFD simulation using the **Models** group of the **Physics** ribbon tab.
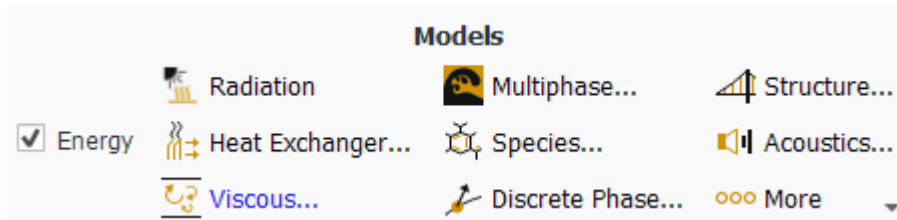
   a. Enable heat transfer by activating the energy equation.

      In the **Physics** ribbon tab, enable **Energy** (**Models** group).

      ▸▸◂◂ **Physics → Models → Energy**

b. Enable the turbulence model.

Physics → Models → Viscous...
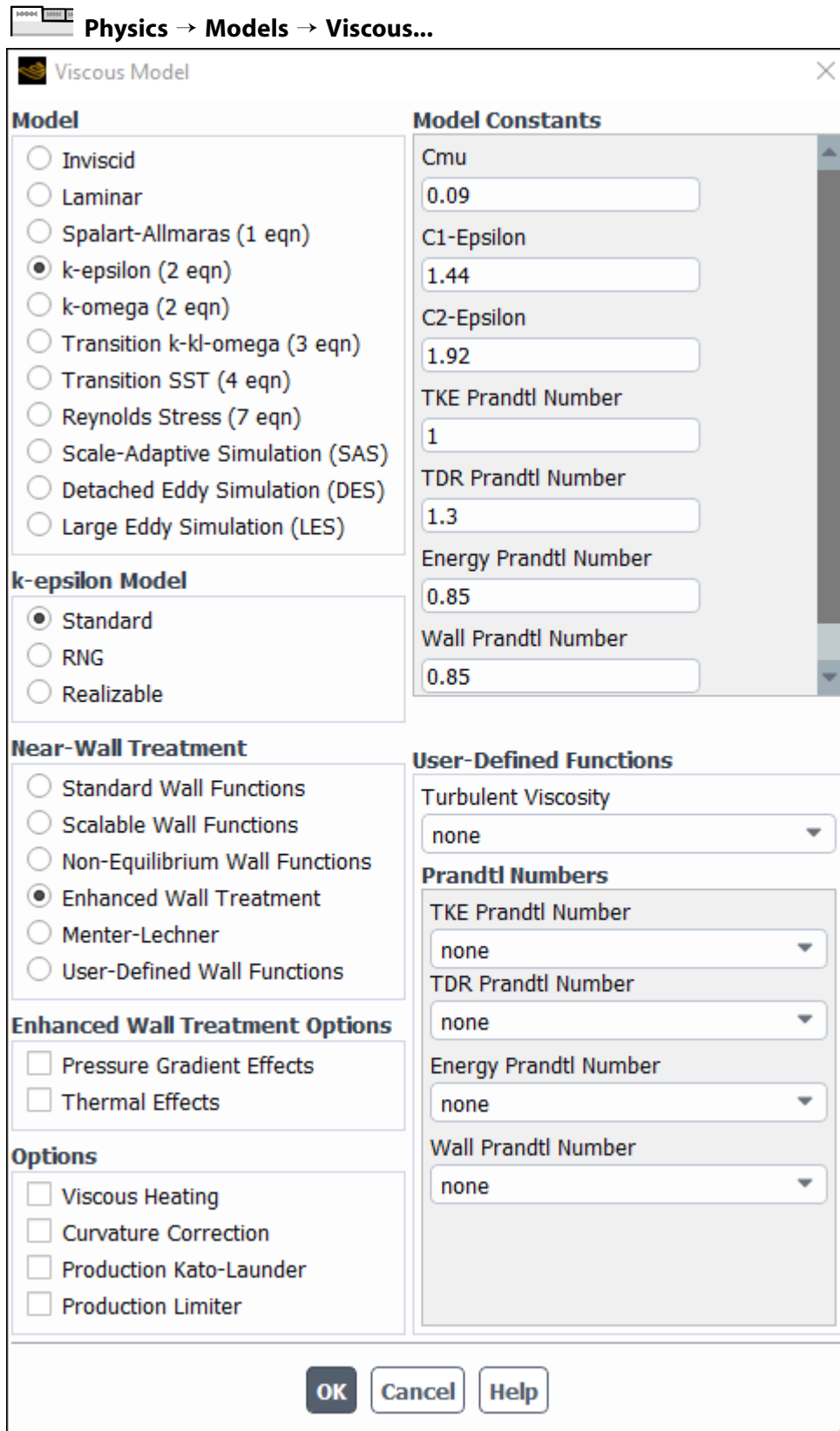
i. Select the **k-epsilon (2 eqn)** from the **Model** group box.

ii. Select **Enhanced Wall Treatment** from the **Near-Wall Treatment** group box.

iii. Click **OK** to retain the other default settings and close the **Viscous Model** dialog box.

Note that the **Viscous...** label in the ribbon is displayed in blue to indicate that the Viscous model is enabled.

3. Define a heat source cell zone condition for the evaporator volume.

⌗ **Physics** → **Zones** → **Cell Zones**

> **Note:**
>
> All cell zones defined in your simulation are listed in the **Cell Zone Conditions** task page and under the **Setup/Cell Zone Conditions** tree branch.

a. In the **Cell Zone Conditions** task page, under the **Zone** list, select **fluid-evaporator** and click **Edit...** to open the **Fluid** dialog box.
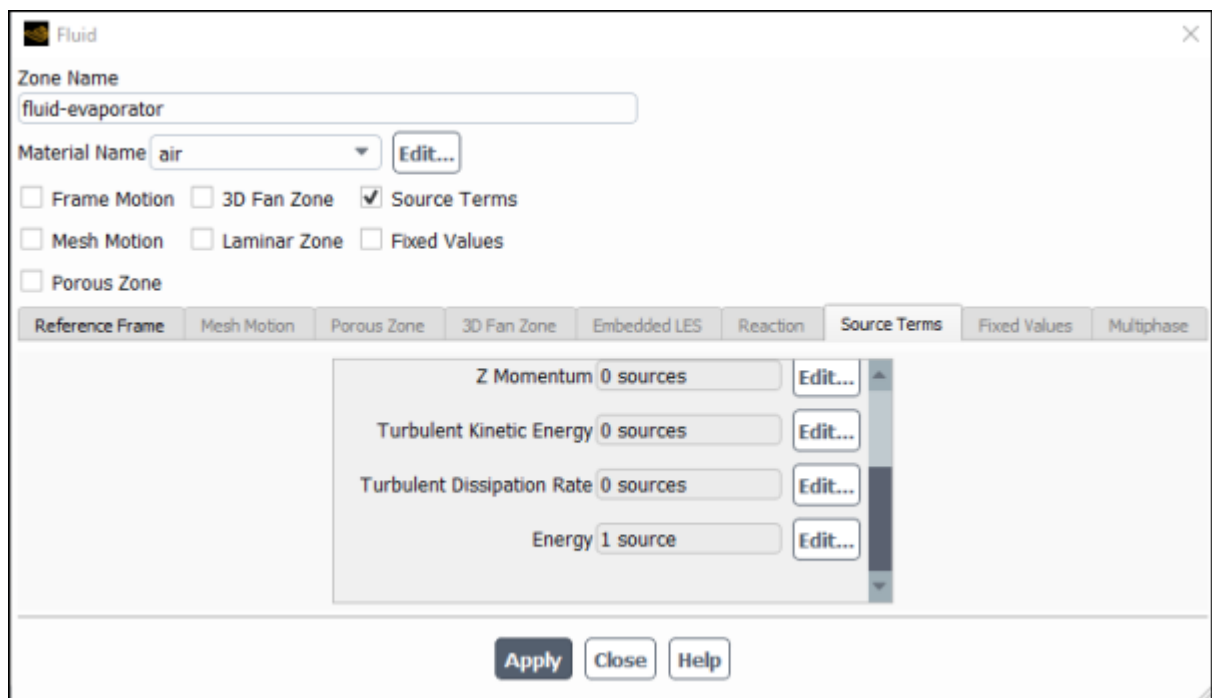
b. In the **Fluid** dialog box, enable **Source Terms**.

c. In the **Source Terms** tab, click the **Edit...** button next to **Energy**.
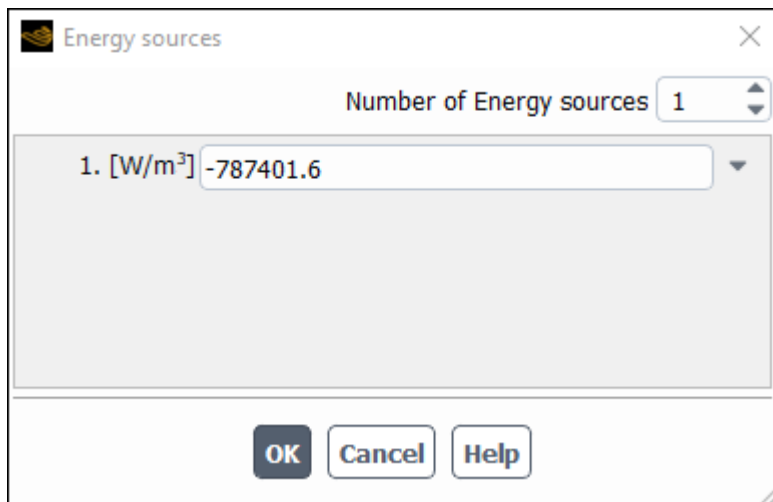
d. In the **Energy sources** dialog box, change the **Number of Energy sources** to 1.

e. For the new energy source, select **constant** from the drop-down list, and enter `-787401.6` W/m^3 — based on the evaporator load (200 W) divided by the evaporator volume (0.000254 m^3) that was computed earlier.

f. Click **OK** to close the **Energy sources** dialog box.

g. Click **Apply** and close the **Fluid** dialog box.

2.4.4. Defining Input Parameters in Ansys Fluent

You have now started setting up the CFD analysis using Ansys Fluent. In this step, you will define boundary conditions and input parameters for the velocity inlet.

1. Define an input parameter called `in_velocity` for the velocity at the inlet boundary.

a. In the **Physics** tab, click **Boundaries** (**Zones** group).

 Physics → Zones → Boundaries

This opens the **Boundary Conditions** task page.

> **Note:**
>
> All boundaries defined in the case are also displayed under the **Setup/Boundary Conditions** tree branch.

b. In the **Boundary Conditions** task page, click the **Toggle Tree View** button (in the upper right corner), and under the **Group By** category, select **Zone Type**. This displays boundary zones grouped by zone type.

c. Under the **Inlet** zone type, double-click **inlet-air**.

d. In the **Velocity Inlet** dialog box, from the **Velocity Magnitude** drop-down list, select **New Input Parameter…**.

*This displays the **Parameter Expression** dialog box.*
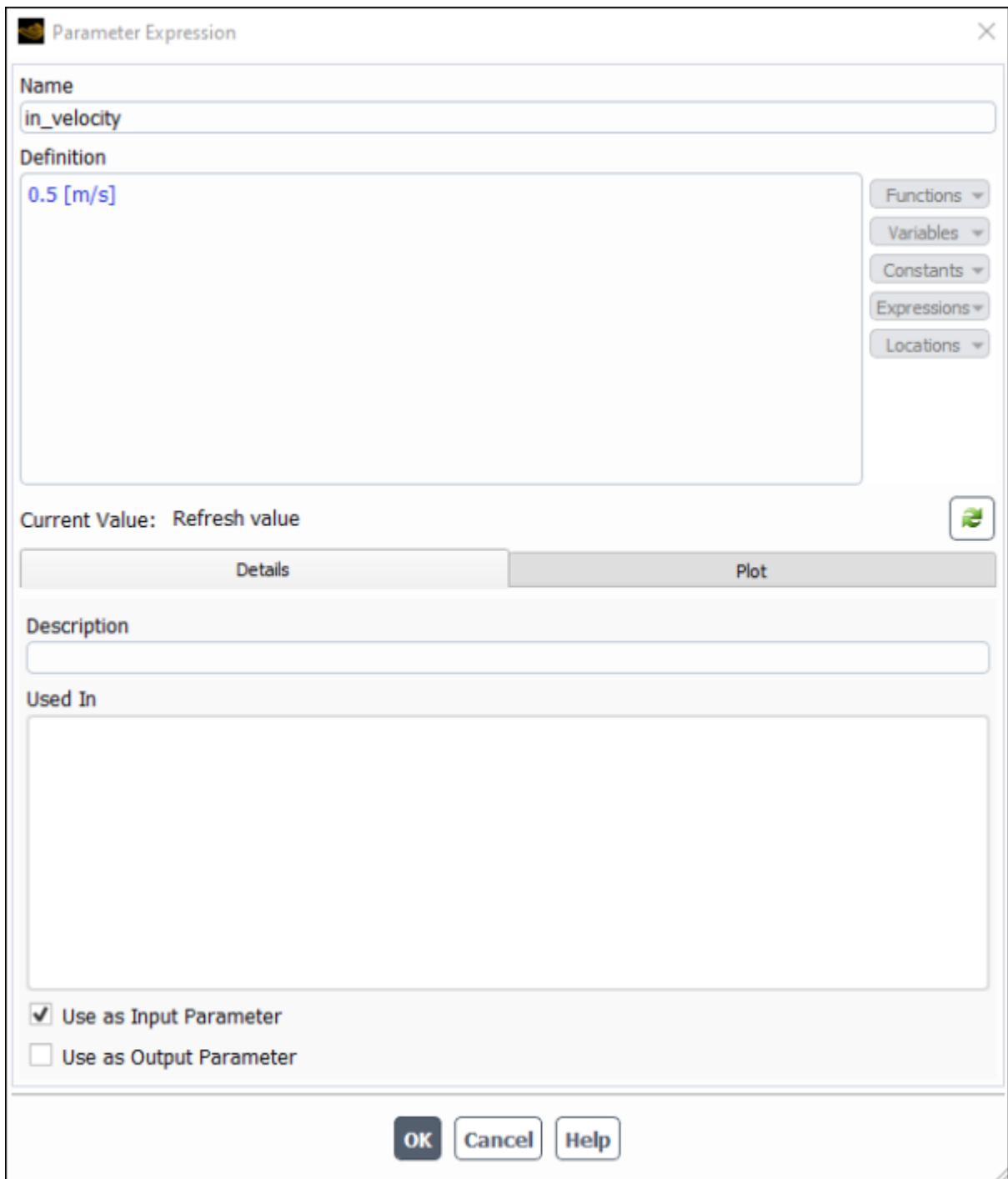
e. Enter `in_velocity` for the **Name**, and enter `0.5 [m s^-1]` for **Definition**.

f. Click **OK** to close the **Parameter Expression** dialog box.

g. Under the **Turbulence** group box, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.

h. Retain the value of 5 % for **Turbulent Intensity**.

i. Enter `0.061` for **Hydraulic Diameter (m)**.

2. Define an input parameter called `in_temp` for the temperature at the inlet boundary.
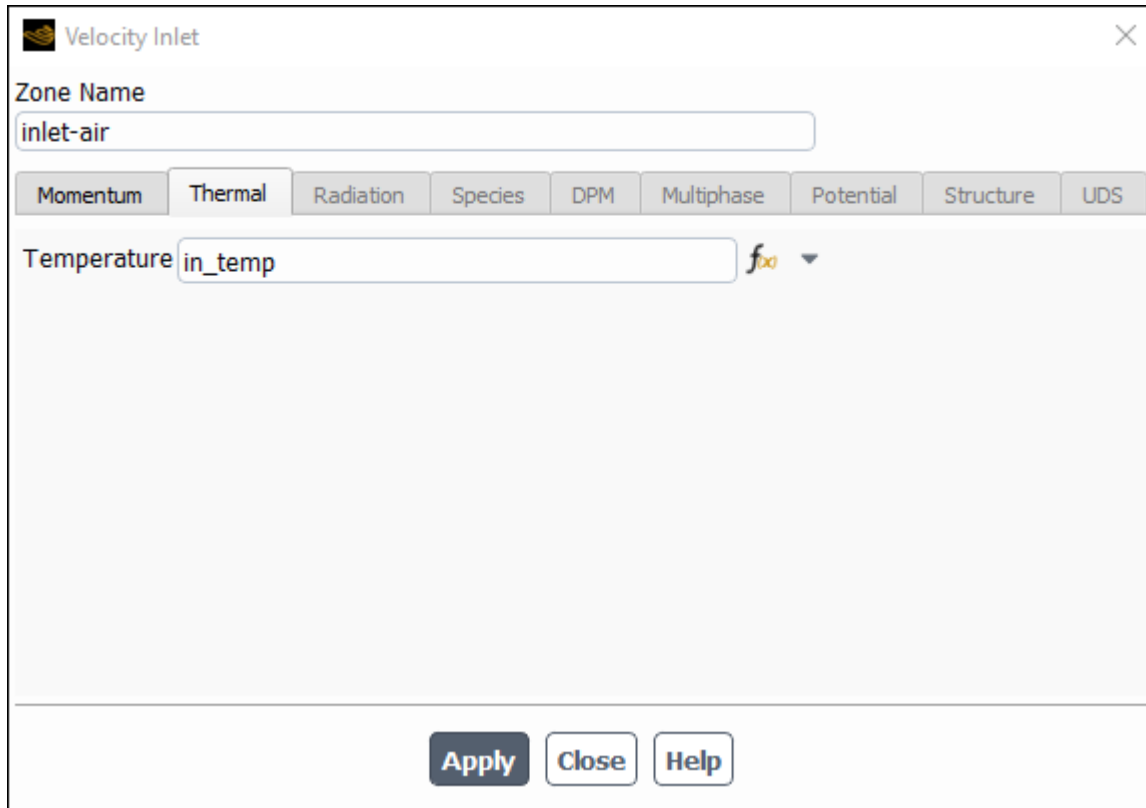
a. In the **Thermal** tab of the **Velocity Inlet** dialog box, select **New Input Parameter...** from the **Temperature** drop-down list.

b. Enter `in_temp` for the **Name** and enter `310 [K]` for the **Definition** in the **Parameter Expression** dialog box.

c. Click **OK** to close the **Parameter Expression** dialog box.

d. Click **Apply** and **Close** the **Velocity Inlet** dialog box.

3. Review all of the input parameters that you have defined in Ansys Fluent under the **Parameters & Customization/Parameters** tree branch.

Parameters & Customization → **Parameters** → **Input Parameters**

Figure 2.12: The Input Parameters Sub-Branch in Ansys Fluent
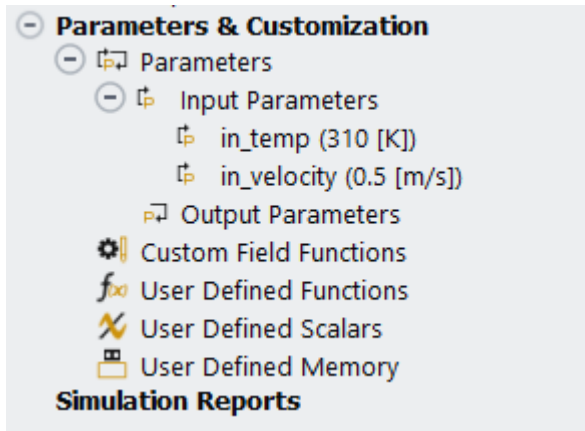
These parameters are passed to Ansys Fluent component system in Ansys Workbench and are available for editing in Ansys Workbench (see Figure 2.13: The Parameters View in Ansys Workbench (p. 88)).

Figure 2.13: The Parameters View in Ansys Workbench

		A	B	C	D
		ID	Parameter Name	Value	Unit
1		ID	Parameter Name	Value	Unit
2		Input Parameters			
3		Fluid Flow (FLUENT) (A1)			
4		P1	hcpos	25	
5		P2	ftpos	25	
6		P3	wsfpos	90	
7		P7	in_velocity	0.5	m s^-1
8		P8	in_temp	310	K
9		P4	input_hcpos	15	
10		P5	input_ftpos	25	
11		P6	input_wspos	90	
*		New input parameter	New name	New expression	
13		Output Parameters			
*		New output parameter		New expression	
15		Charts			

4. Set the turbulence parameters for backflow at the front outlets and foot outlets.

a. In the **Boundary Conditions** task page, type `outlet` in the **Zone** filter text entry field. Note that as you type, the names of the boundary zones beginning with the characters you entered appear in the boundary condition zone list.

Note:

- The search string can include wildcards. For example, entering `*let*` will display all zone names containing `let`, such as `inlet` and `outlet`.

- To display all zones again, click the red **X** icon in the **Zone** filter.

---

b. Double-click **outlet-front-mid**.

c. In the **Pressure Outlet** dialog box, under the **Turbulence** group box, select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.

d. Retain the value of 5 for **Backflow Turbulent Intensity (%)**.

e. Enter 0.044 for **Backflow Hydraulic Diameter (m)**.

These values will only be used if reversed flow occurs at the outlets. It is a good idea to set reasonable values to prevent adverse convergence behavior if backflow occurs during the calculation.

f. Click **Apply** and **Close** the **Pressure Outlet** dialog box.

g. Copy the boundary conditions from **outlet-front-mid** to the other front outlet.

⌨ Setup → **Boundary Conditions** → **outlet-front-mid** 🖱 **Copy...**
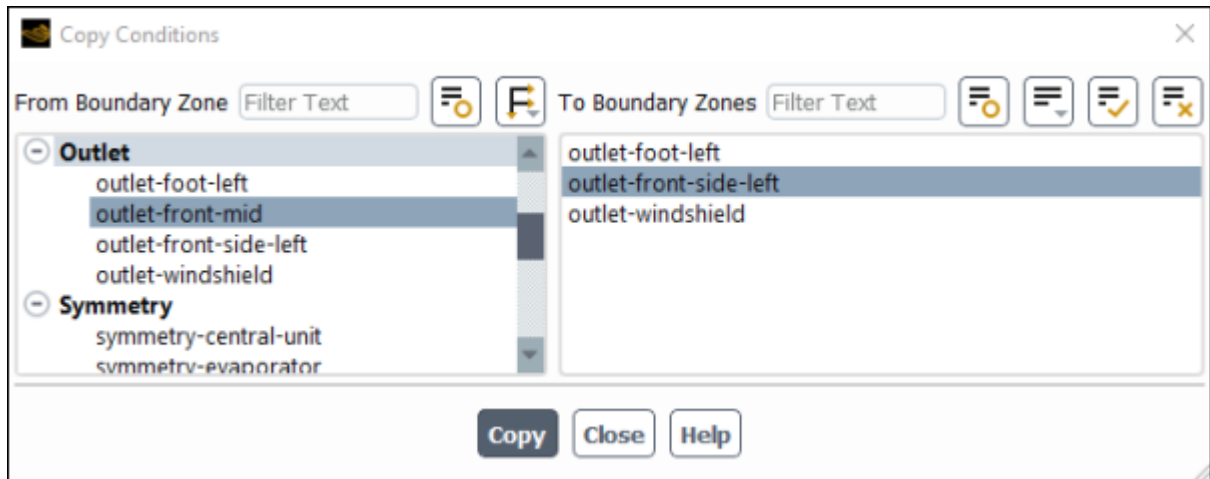
i. Confirm that **outlet-front-mid** is selected in the **From Boundary Zone** selection list.

ii. Select **outlet-front-side-left** in the **To Boundary Zones** selection list.

iii. Click **Copy** to copy the boundary conditions.

Fluent will display a dialog box asking you to confirm that you want to copy the boundary conditions.

iv. Click **OK** to confirm.

v. Close the **Copy Conditions** dialog box.

h. In a similar manner, set the backflow turbulence conditions for **outlet-foot-left** using the values in the following table:

Parameter	Value
Specification Method	Intensity and Hydraulic Diameter
Backflow Turbulent Intensity (%)	5
Backflow Hydraulic Diameter (m)	0.052

2.4.5. Solving

In the steps that follow, you will set up and run the calculation using the **Solution** ribbon tab.

Note:

You can also use the task pages listed under the **Solution** branch in the tree to perform solution-related activities.

---

1. Set the Solution Methods.

 Solution → **Solution** → **Methods...**

This will open the **Solution Methods** task page.
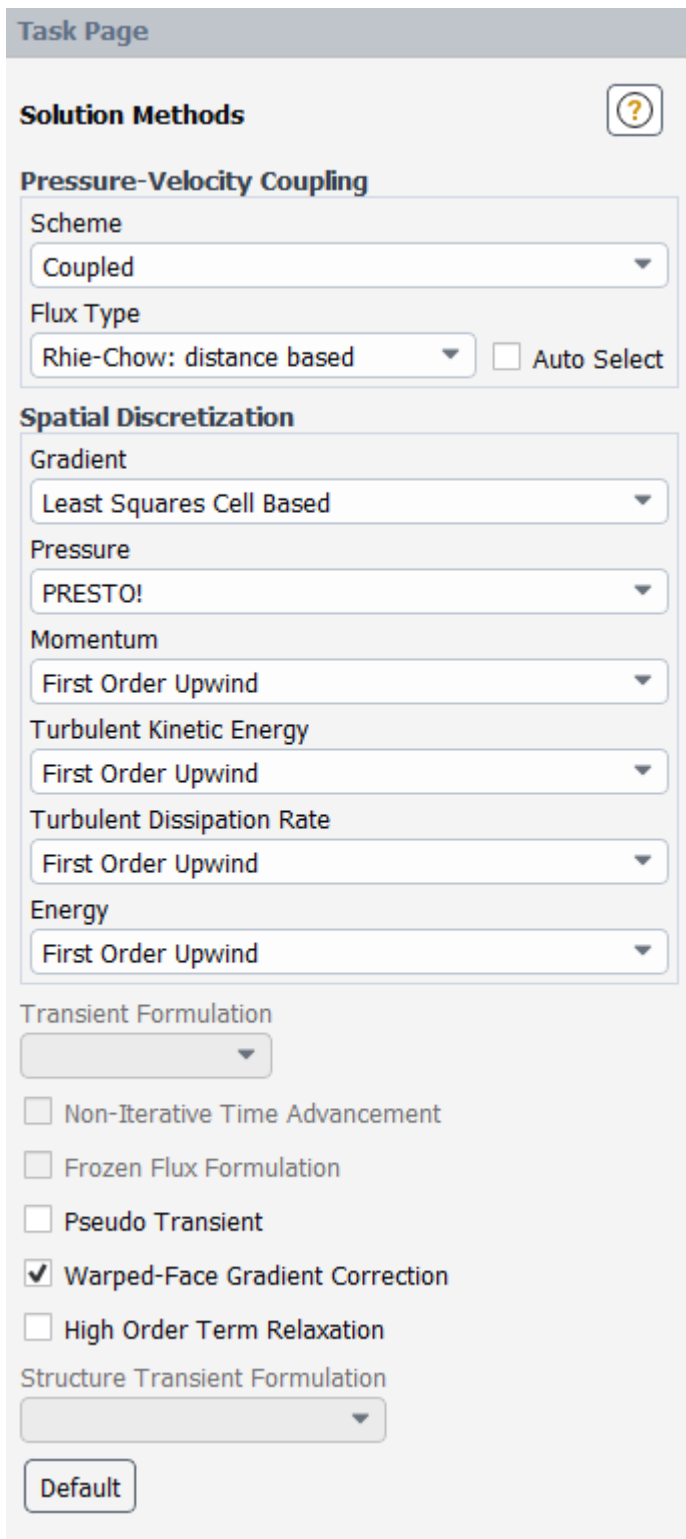
a. From the **Scheme** drop-down list, ensure that **Coupled** is selected.

b. In the **Spatial Discretization** group box, configure the following settings:

Setting	Value
Pressure	PRESTO!

Setting	Value
Momentum	First Order Upwind
Energy	First Order Upwind

This tutorial is primarily intended to demonstrate the use of parameterization and design points when running Fluent from Workbench. Therefore, you will run a simplified analysis using first order discretization, which will yield faster convergence. These settings were chosen to speed up solution time for this tutorial. Usually, for cases like this, we would recommend higher order discretization settings to be set for all flow equations to ensure improved results accuracy. The second-order scheme will provide a more accurate solution.

   c. Ensure that **Warped-Faced Gradient Correction** is selected.

   d. Ensure that **Pseudo Transient** is not selected.

2. Initialize the flow field using the **Initialization** group of the **Solution** ribbon tab.

    Solution → Initialization
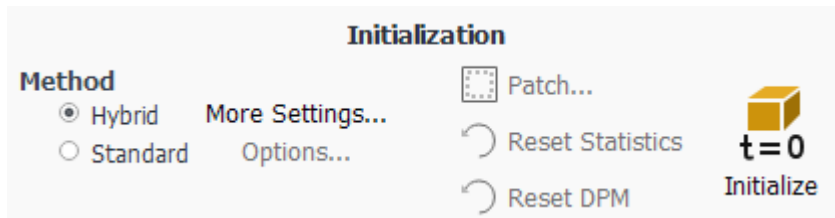
   a. Retain the default selection of **Hybrid Initialization**.

   b. Click the **Initialize** button.

3. Run the simulation in Ansys Fluent from the **Run Calculation** group of the **Solution** tab.

    Solution → Run Calculation
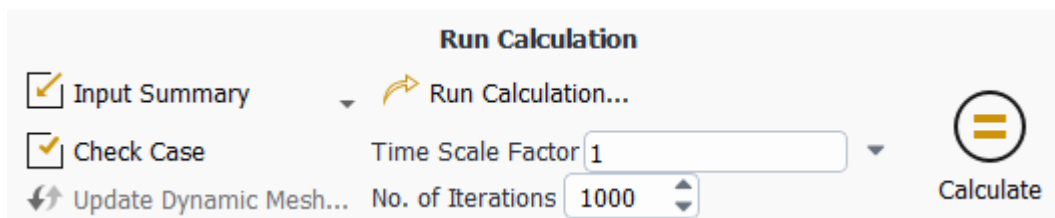
   a. Enter `1000` for **No. of Iterations**

   b. Click the **Calculate** button.

Throughout the calculation, Fluent displays a warning in the console regarding reversed flow at the outlets. This behavior is expected in this case since air is redirected to the outlets, creating small regions of recirculation.

4. Close Fluent.

⊨⊨ **File → Close Fluent**

5. Save the project in Ansys Workbench.

    File → Save

2.4.6. Postprocessing and Setting the Output Parameters in Ansys CFD-Post

In this step, you will visualize the results of your CFD simulation using Ansys CFD-Post. You will plot vectors that are colored by pressure, velocity, and temperature, on a plane within the geometry. In addition, you will create output parameters within Ansys CFD-Post for later use in Ansys Workbench.

In the Ansys Workbench **Project Schematic**, double-click the **Results** cell in the Ansys Fluent fluid flow analysis system to start CFD-Post. You can also right-click the **Results** cell to display the context menu where you can select the **Edit...** option.

The CFD-Post application appears with the automotive HVAC geometry already loaded and displayed in outline mode. Note that Ansys Fluent results (that is, the case and data files) are also automatically loaded into CFD-Post.

Figure 2.14: The Automotive HVAC Geometry Loaded into CFD-Post
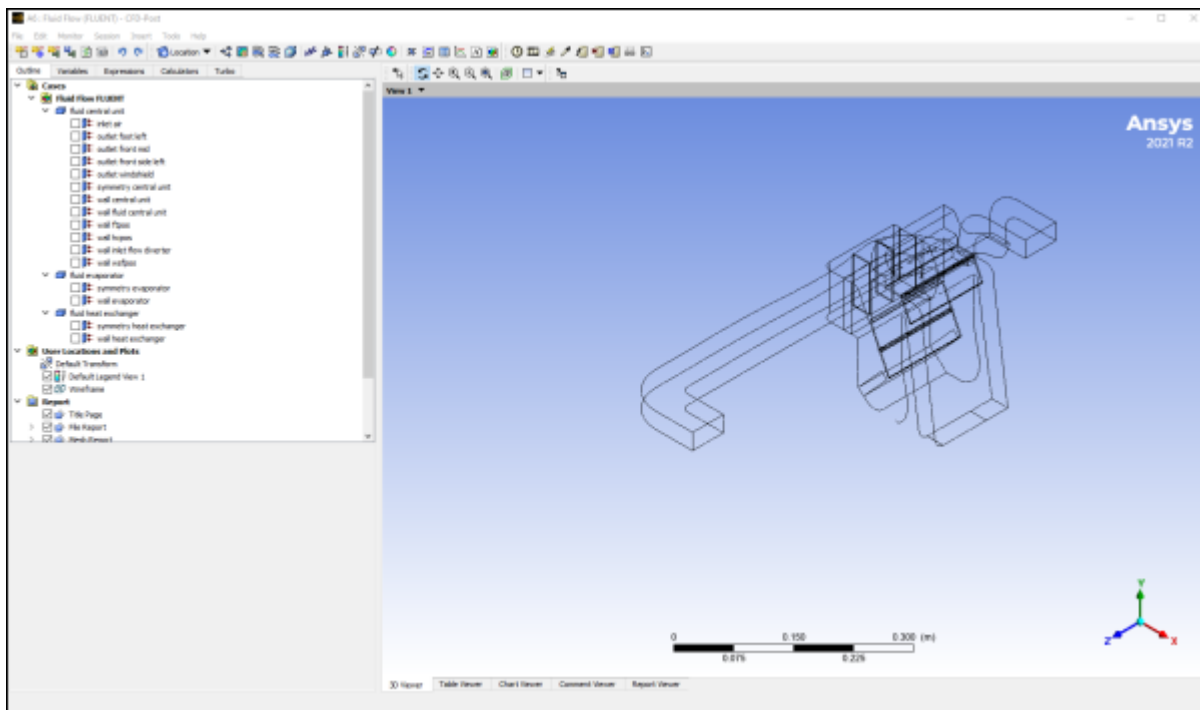
1. Plot vectors colored by pressure.

    a. From the main menu, select **Insert → Vector** or click ⇨⇔ in the Ansys CFD-Post toolbar.

        *This displays the **Insert Vector** dialog box.*

    b. Keep the default name of **Vector 1** by clicking **OK**.

    c. In the **Details** view for **Vector 1**, under the **Geometry** tab, configure the following settings.
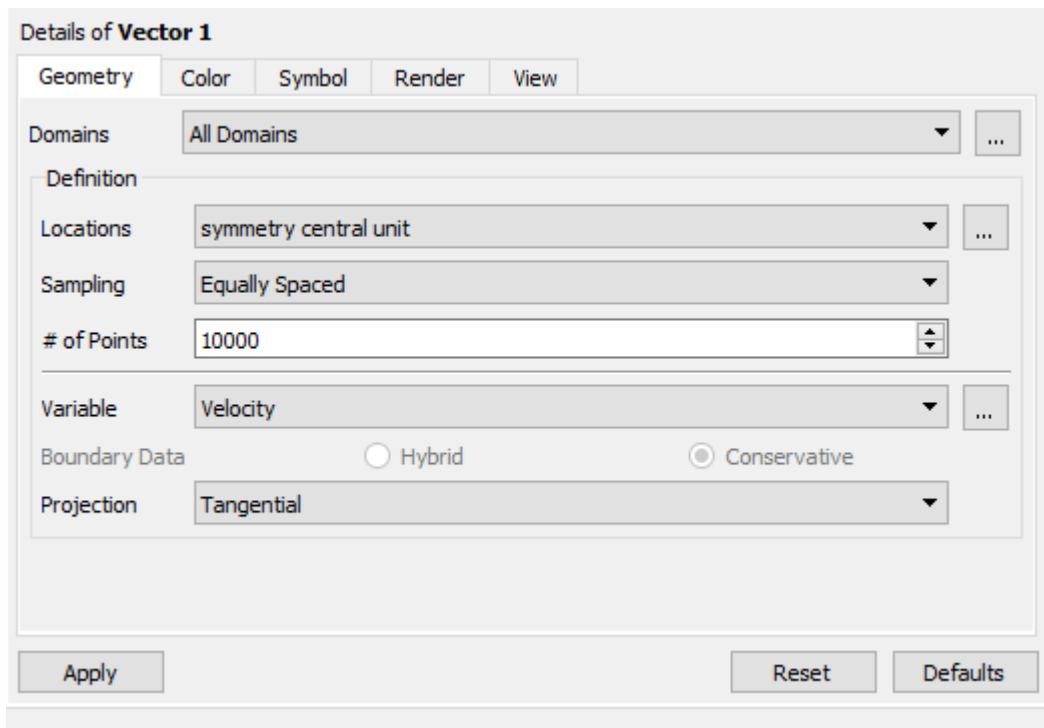
i. Ensure **All Domains** is selected from the **Domains** drop-down list.

ii. From the **Locations** drop-down list, select **symmetry central unit**.

iii. From the **Sampling** drop-down list, select **Equally Spaced**.

iv. Set the **# of Points** to `10000`.

v. From the **Projection** drop-down list, select **Tangential**.

d. In the **Color** tab, configure the following settings.

Details of **Vector 1**

| Geometry | Color | Symbol | Render | View |

Mode — Variable

Variable — Pressure

Range — Global

Min — unknown

Max — unknown

Boundary Data ○ Hybrid ◉ Conservative

Color Scale — Linear

Color Map — Default (Rainbow)

Undef. Color

Apply   Reset   Defaults
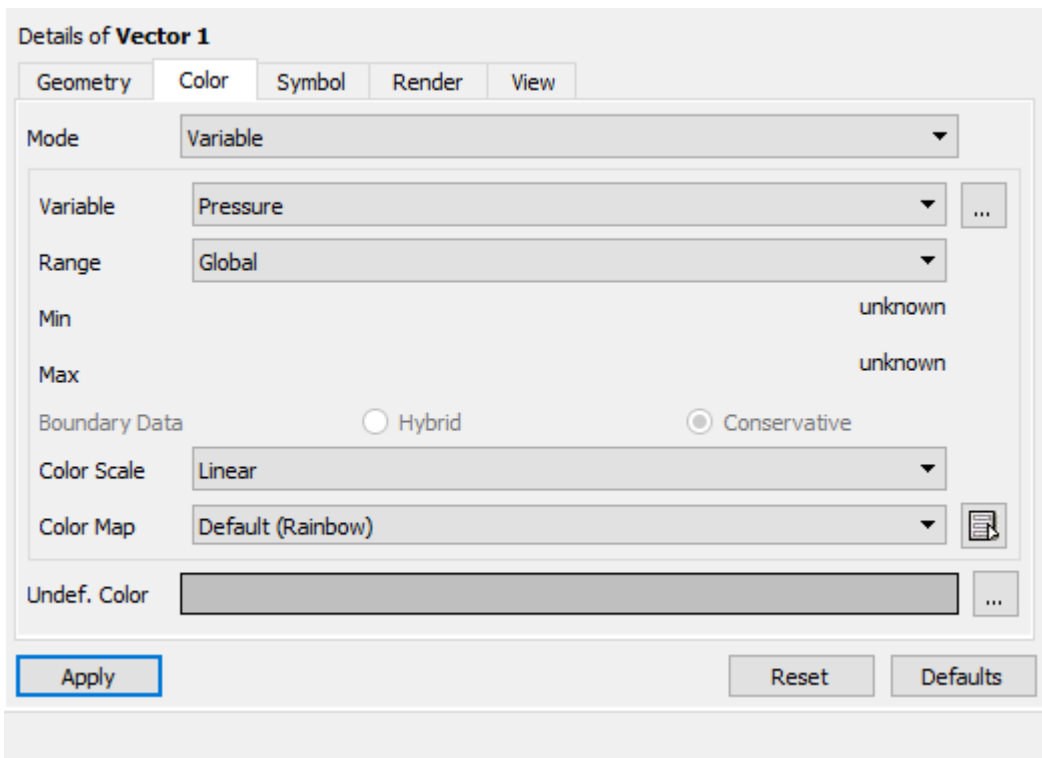
i. From the **Mode** drop-down list, select **Variable**.

ii. From the **Variable** drop-down list, select **Pressure**.

e. In the **Symbol** tab, configure the following settings.

Details of **Vector 1**

| Geometry | Color | Symbol | Render | View |

Symbol — Line Arrow

Symbol Size — 0.5

☐ Normalize Symbols

Apply   Reset   Defaults
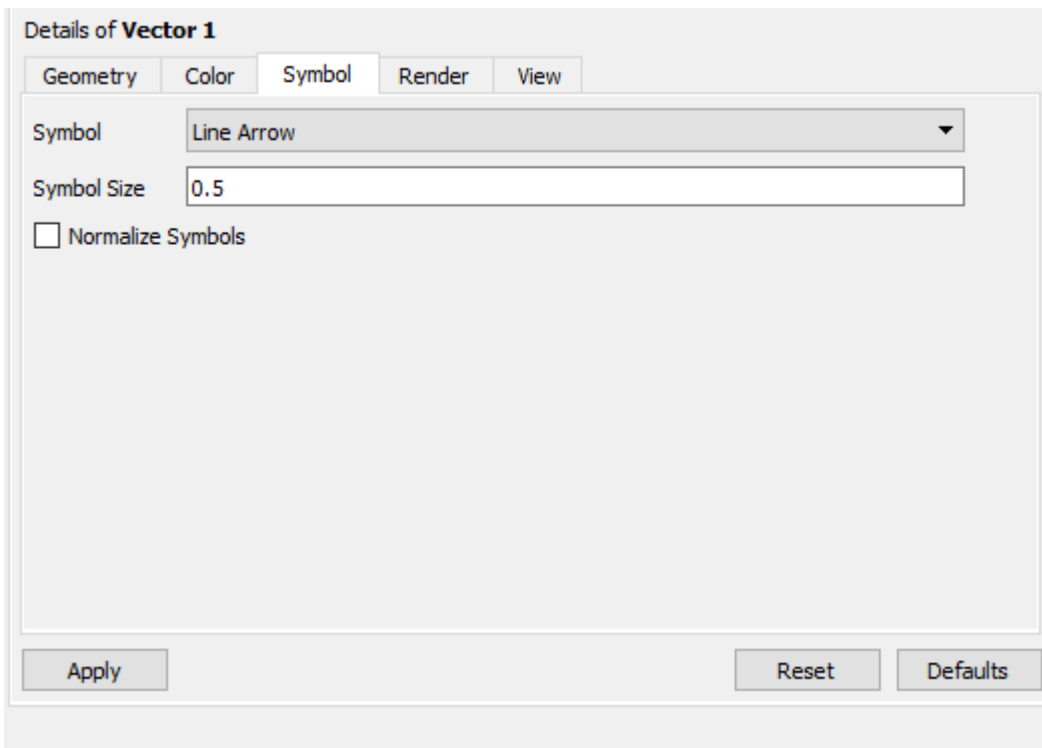
i. Set the **Symbol Size** to 0.5.

f. Click **Apply**.

Vector 1 appears under **User Locations and Plots** in the **Outline** tree view.

*In the graphics display window, note that **symmetry-central-unit** shows the vectors colored by pressure. Use the controls in CFD-Post to rotate the geometry (for example, clicking the dark blue axis in the axis triad of the graphics window). Zoom in to the view as shown in Figure 2.15: Vectors Colored by Pressure (p. 97).*

> **Note:**
>
> To better visualize the vector display, you can deselect the **Wireframe** view option under **User Locations and Plots** in the **Outline** tree view.

Figure 2.15: Vectors Colored by Pressure
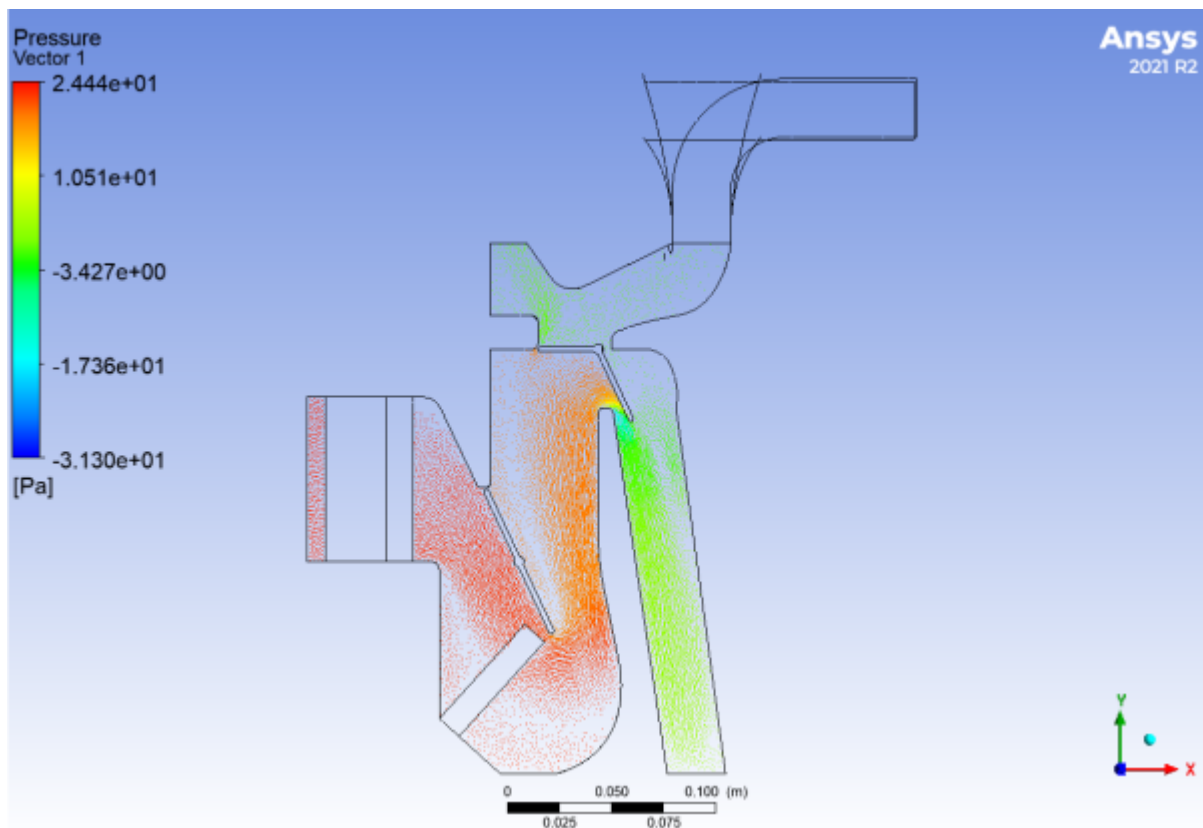
2. Plot vectors colored by velocity.

   a. In the **Details** view for **Vector 1**, under the **Color** tab, configure the following settings.

      i. Select **Velocity** from the **Variable** drop-down list.

      ii. Click **Apply**.

      The velocity vector plot appears on the **symmetry-central-unit** symmetry plane.

Figure 2.16: Vectors Colored by Velocity
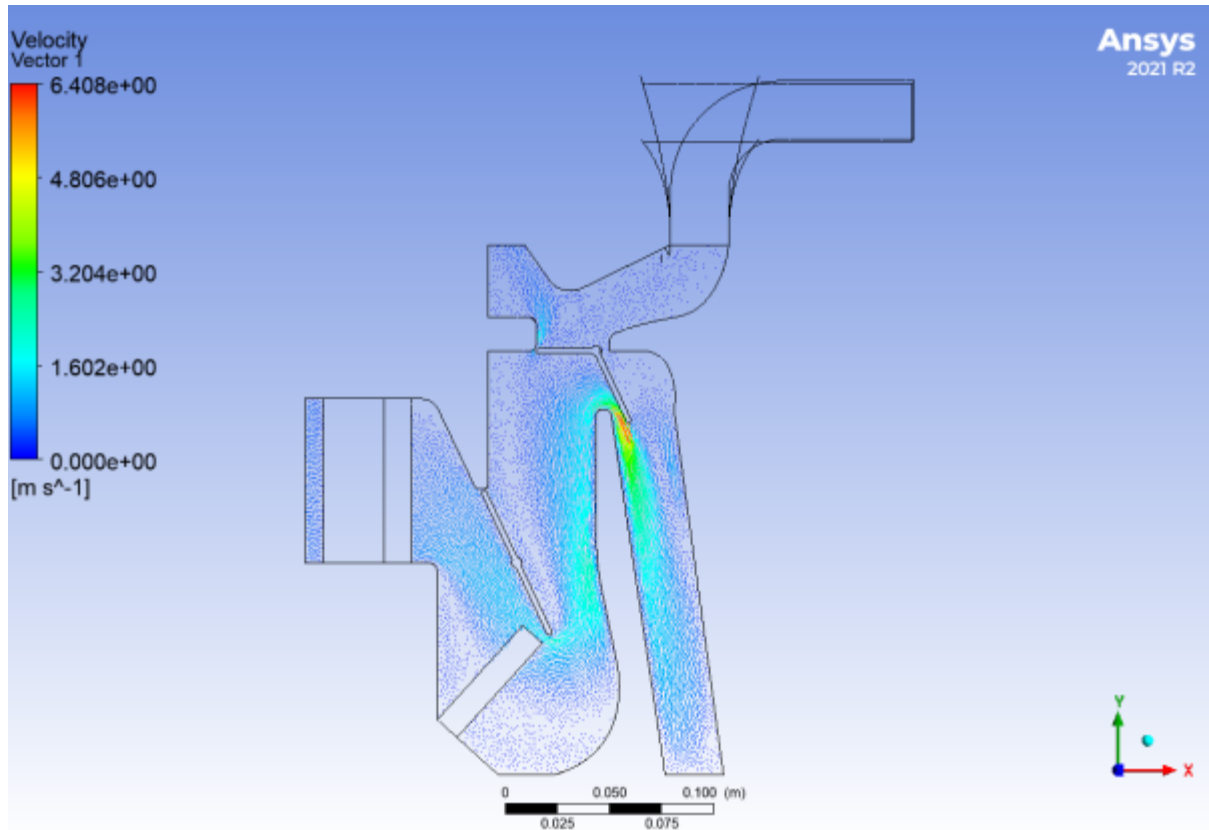
3. Plot vectors colored by temperature.

    a. In the **Details** view for **Vector 1**, under the **Color** tab, configure the following settings.

        i. Select **Temperature** from the **Variable** drop-down list.

        ii. Select **User Specified** from the **Range** drop-down list.

        iii. Enter 273 K for the **Min** temperature value.

        iv. Enter 310 K for the **Max** temperature value.

        v. Click **Apply**.

            The user-specified range is selected much narrower than the Global and Local ranges in order to better show the variation.

Figure 2.17: Vectors Colored by Temperature
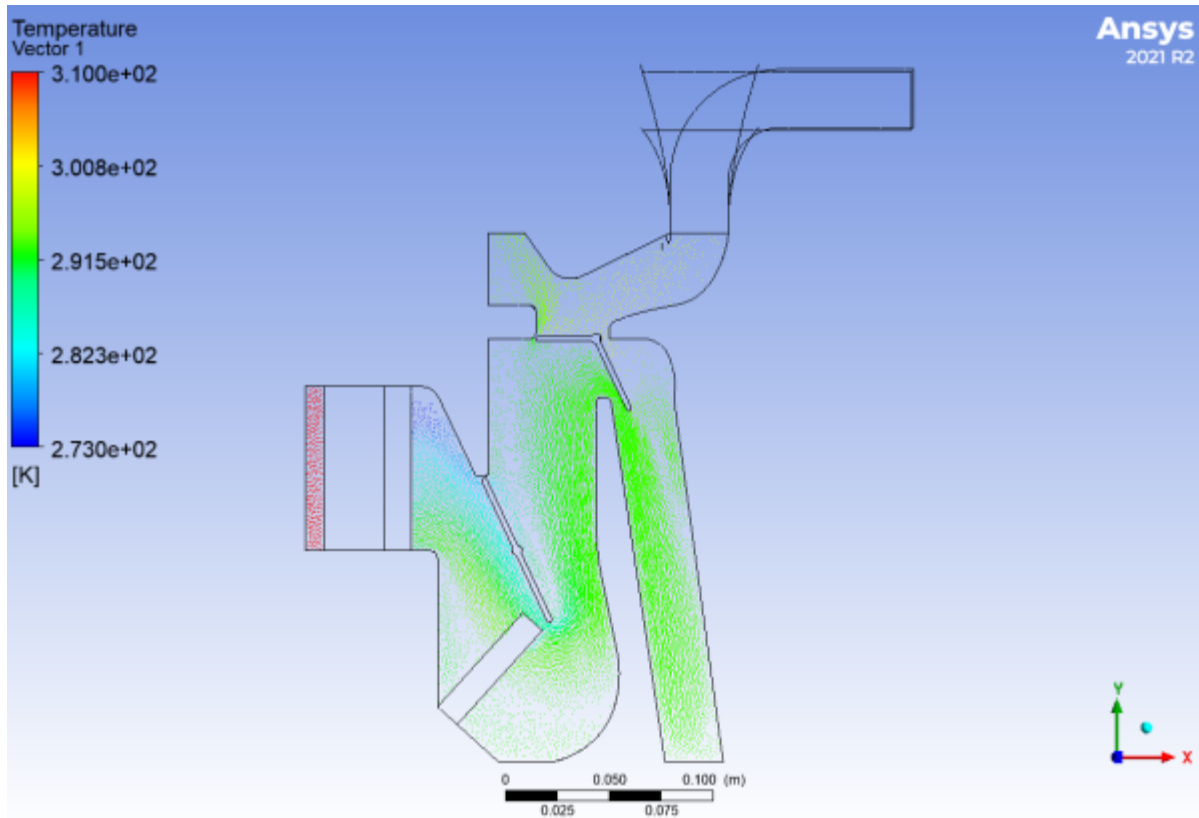
Note the orientation of the various valves and how they impact the flow field. Later in this tutorial, you will change these valve angles to see how the flow field changes.

4. Create two surface groups.

   Surface groups are collections of surface locations in CFD-Post. In this tutorial, two surface groups are created in CFD-Post that will represent all of the outlets and all of the front outlets. Once created, specific commands (or expressions) will be applied to these groups in order to calculate a particular numerical value at that surface.

   a. Create a surface group consisting of all outlets.

      i. With the **Outline** tree view open in the CFD-Post tree view, open the **Insert Surface Group** dialog box.

         Insert → Location → Surface Group
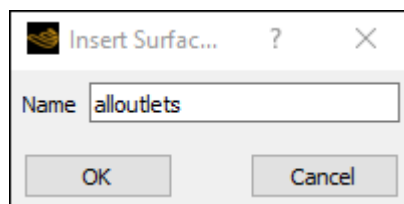
ii. Enter `alloutlets` for the **Name** of the surface group, and click **OK** to close the **Insert Surface Group** dialog box.

iii. In the **Details** view for the `alloutlets` surface group, in the **Geometry** tab, click the ellipsis icon [...] next to **Locations** to display the **Location Selector** dialog box.
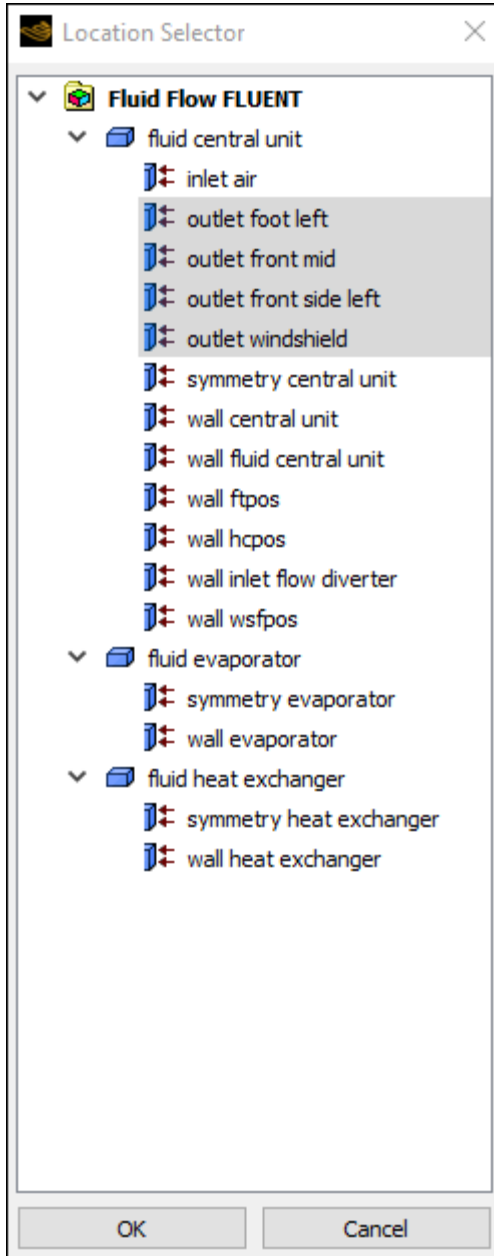
iv. Select all of the outlet surfaces (`outlet foot left`, `outlet front mid`, `outlet front side left`, and `outlet windshield`) in the **Location Selector** dialog box (hold **Ctrl** for multiple selection) and click **OK**.

v. Click **Apply** in the **Details** view for the new surface group.

`alloutlets` appears under **User Locations and Plots** in the **Outline** tree view.

b. Create a surface group for the front outlets.

Perform the same steps as described above to create a surface group called `frontoutlets` with locations for the front outlets (`outlet front mid` and `outlet front side left`).

5. Create expressions in CFD-Post and mark them as Ansys Workbench output parameters.

In this tutorial, programmatic commands or expressions are written to obtain numerical values for the mass flow rate from all outlets, as well as at the front outlets, windshield, and foot outlets. The surface groups you defined earlier are used to write the expressions.

a. Create an expression for the mass flow from all outlets.

    i. With the **Expressions** tab open in the CFD-Post tree view, open the **Insert Expression** dialog box.

    Insert → Expression

    ii. Enter `floutfront` for the **Name** of the expression and click **OK** to close the **Insert Expression** dialog box.
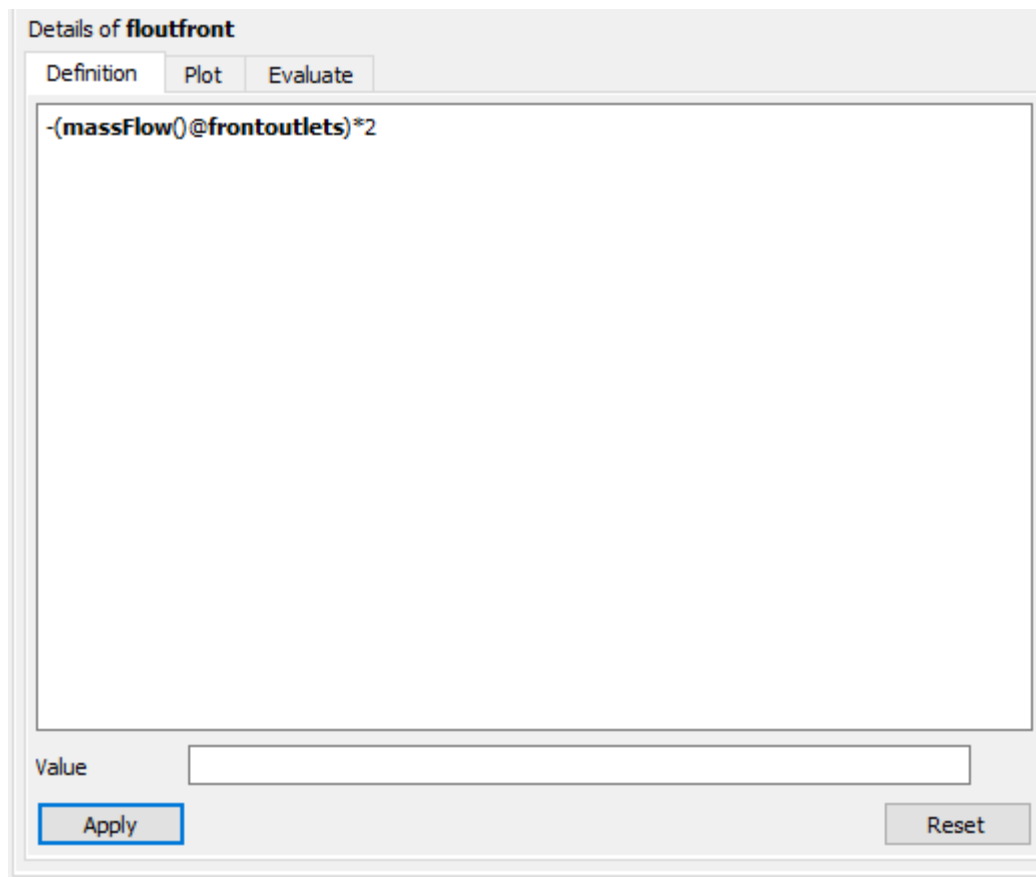
    iii. In the **Details** view for the new expression, enter the following in the **Definition** tab.

    ```
    -(massFlow()@frontoutlets)*2
    ```

*The sign convention for **massFlow()** is such that a positive value represents flow into the domain and a negative value represents flow out of the domain. Since you are defining an expression for outflow from the ducts, you use the negative of the **massFlow()** result in the definition of the expression.*

iv. Click **Apply** to obtain a **Value** for the expression.

*Note the new addition in the list of expressions in the **Expressions** tab in CFD-Post.*

In this case, there is a small net backflow into the front ducts.

v. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu. A small "P" with a right-pointing arrow appears on the expression's icon.

b. Create an expression for the mass flow from the wind shield.

   i. Perform the same steps as described above to create an expression called `floutwindshield` with the following definition:

```
-(massFlow()@outlet windshield)*2
```

   ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.

c. Create an expression for the mass flow from the foot outlets.

   i. Perform the same steps as described above to create an expression called `floutfoot` with the following definition:

```
-(massFlow()@outlet foot left)*2
```

   ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.

d. Create an expression for the mass weighted average outlet temperature.

   i. Perform the same steps as described above to create an expression called `outlettemp` with the following definition:

```
massFlowAveAbs(Temperature)@alloutlets
```

   ii. Right-click the new expression and select **Use as Workbench Output Parameter** from the context menu.

6. Close Ansys CFD-Post.

   In the main menu, select **File** → **Close CFD-POST** to return to Ansys Workbench.

7. In the **Outline of All Parameters** view of the **Parameter Set** tab (double-click **Parameter Set**), review the newly-added output parameters that you specified in Ansys CFD-Post and when finished, click the **Project** tab to return to the **Project Schematic**.

8. If any of the cells in the analysis system require attention, update the project by clicking the **Update Project** button in the Ansys Workbench toolbar.

9. Optionally, review the list of files generated by Ansys Workbench. If the **Files** view is not open, select **View** → **Files** from the main menu.

You will notice additional files associated with the latest solution as well as those generated by CFD-Post.

Figure 2.18: The Updated Project Loaded into Ansys Workbench Displaying the Files View
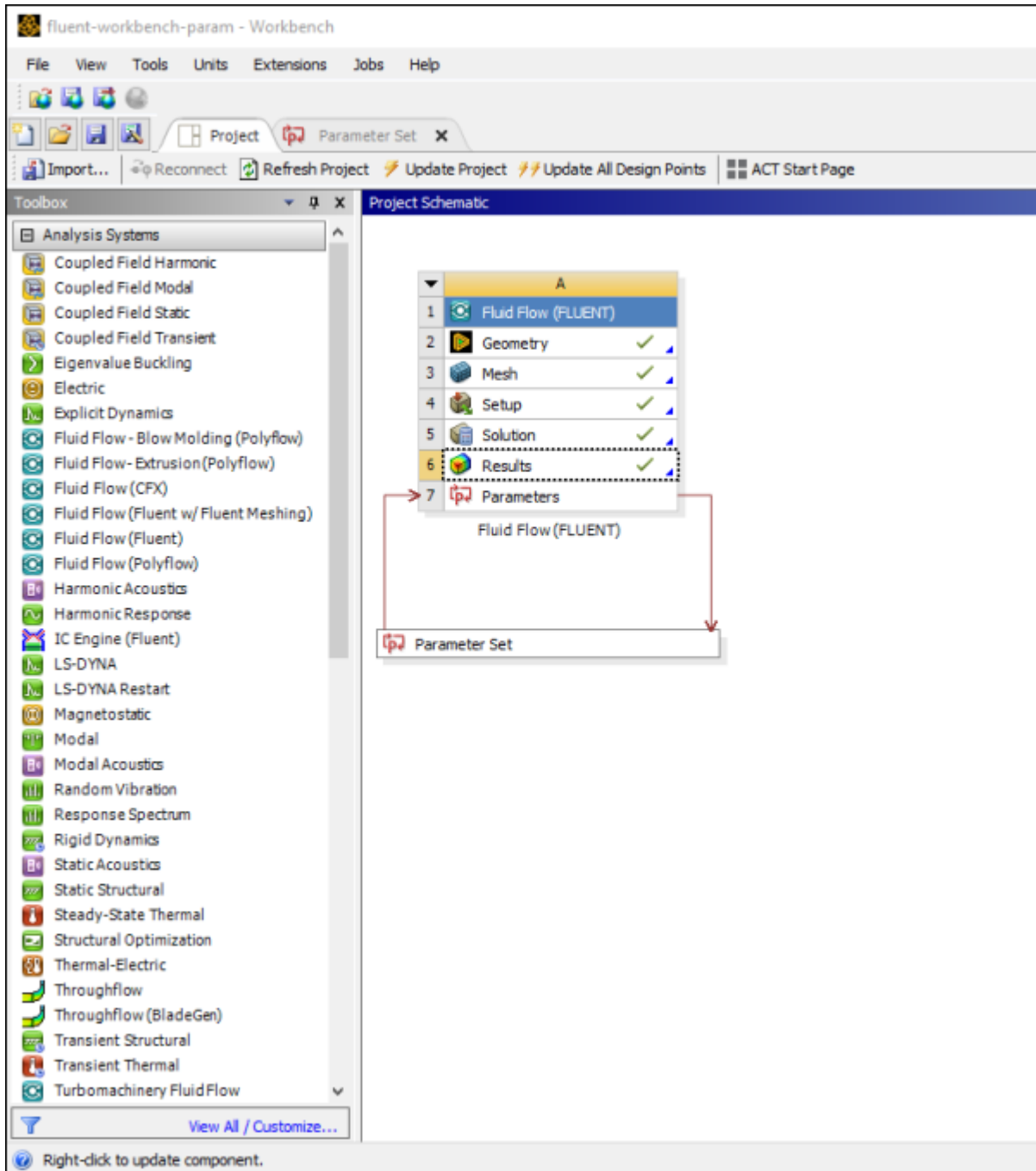
10. Save the project in Ansys Workbench.

In the main menu, select **File** → **Save**

Note:

You can also select the **Save Project** option from the CFD-Post **File** ribbon tab.

2.4.7. Creating Additional Design Points in Ansys Workbench

Parameters and design points are tools that allow you to analyze and explore a project by giving you the ability to run optimization and what-if scenarios. Design points are based on sets of parameter values. When you define input and output parameters in your Ansys Workbench project, you are essentially working with a design point. To perform optimization and what-if scenarios, you create multiple design points based on your original project. In this step, you will create additional design points for your project where you will be able to perform a comparison of your results by manipulating input parameters (such as the angles of the various valves within the automotive HVAC geometry). Ansys Workbench provides a Table of Design Points to make creating and manipulating design points more convenient.

1. Open the Table of Design Points.

    a. In the Project Schematic, double-click the **Parameter Set** bus bar to open the Table of Design Points view. If the table is not visible, select **Table** from the **View** menu in Ansys Workbench.

    View → **Table**

    The table of design points initially contains the current project as a design point (DP0), along with its corresponding input and output parameter values.

    Figure 2.19: Table of Design Points (with DP0)

	A	B	C	D	E	F	G	H	I
1	Name	P1 - hcpos	P2 - ftpos	P3 - wsfpos	P4 - input_hcpos	P5 - input_ftpos	P6 - input_wspos	P7 - in_velocity	P8 - in_temp
2	Units							m s^-1	K
3	DP 0 (Current)	25	25	90	15	25	90	0.5	310
*									

    From this table, you can create new design points (or duplicate existing design points) and edit them (by varying one or more input parameters) to create separate analyses for future comparison of data.

2. Create a design point (DP1) by duplicating the current design point (DP0).

    a. Right-click the **Current** design point and select **Duplicate Design Point** from the context menu.

    The cells autofill with the values from the **Current** row.

    b. Scroll over to the far right to expose the **Retain** column in the table of design points, and ensure the check box in the row for the duplicated design point **DP 1** (cell N4) is selected.

This allows the data from this new design point to be saved before it is exported for future analysis.

3. Create another design point (DP2) by duplicating the DP1 design point.

   a. Right-click the **DP1** design point and select **Duplicate Design Point** from the context menu.

   Since this is a duplicate of DP1, this design point will also have its data retained.

4. Edit values for the input parameters for DP1 and DP2.

   For DP1 and DP2, edit the values for your input parameters within the Table of Design Points as follows:

	input_hcpos	input_ftpos	input_wsf-pos	in_velocity	in_temp
DP1	45	45	45	0.6	300
DP2	90	60	15	0.7	290

Figure 2.20: Table of Design Points (with DP0, DP1, and DP2 Defined)

For demonstration purposes of this tutorial, in each design point, you are slightly changing the angles of each of the valves, and increasing the inlet velocity and decreasing the inlet temperature. Later, you will see how the results in each case vary.

5. Update all of your design points.

   Click the **Update all Design Points** button in the Ansys Workbench toolbar. Alternatively, you can also select one or more design points, right-click, and select **Update Selected Design Points** from the context menu. Click **OK** to acknowledge the information message notifying you that some open editors may close during the update process. By updating the design points, Ansys Workbench takes the new values of the input parameters for each design point and updates the components of the associated system (for example, the geometry, mesh, settings, solution, and results), as well as any output parameters that have been defined.

   > **Note:**
   >
   > It may take significant time and/or computing resources to re-run the simulations for each design point.

6. Export the design points to separate projects.

   This will allow you to work with calculated data for each design point.

   a. Select the three design points, **DP0**, **DP1**, and **DP2** (hold **Shift** for multiple selection).

   b. Right-click the selected design points and select **Export Selected Design Points**.

     Note the addition of three more Ansys Workbench project files (and their corresponding folders) in your current working directory (`fluent-workbench-param_dp0.wbpj`, `fluent-workbench-param_dp1.wbpj` and `fluent-workbench-param_dp2.wbpj`). You can open each of these projects up separately and examine the results of each parameterized simulation.

Tip:

You can easily access files in your project directory directly from the **Files** view by right-clicking any cell in the corresponding row and selecting **Open Containing Folder** from the menu that opens.

---

7. Inspect the output parameter values in Ansys Workbench.

   Once all design points have been updated, you can use the table of design points to inspect the values of the output parameters you created in CFD-Post (for example, the mass flow parameters at the various outlets: `floutfront`, `floutfoot`, `floutwindshield`, and `outlettemp`). These, and the rest of the output parameters are listed to the far right in the table of design points.

8. Click the **Project** tab, located just above the Ansys Workbench toolbar, to return to the **Project Schematic**.

9. (Optional) View the list of files generated by Ansys Workbench.

   View → Files

   The additional files for the new design points are stored with their respective project files since you exported them.

10. Save the project in its current state in Ansys Workbench.

   In the main menu, select **File → Save**.

11. Quit Ansys Workbench.

   In the main menu, select **File → Exit**.

## 2.4.8. Postprocessing the New Design Points in CFD-Post

In this section, you will open the Ansys Workbench project for each of the design points and inspect the vector plots based on the new results of the simulations.

1. Study the results of the first design point (DP1).

   a. Open the Ansys Workbench project for the first design point (DP1).

In your current working folder, double-click the `fluent-workbench-param_dp1.wbpj` file to open Ansys Workbench.

b. Open CFD-Post by double-clicking the **Results** cell in the Project Schematic for the Fluid Flow (Fluent) analysis system.

c. View the vector plot colored by temperature. Ensure that **Range** in the **Color** tab is set to **User Specified** and the **Min** and **Max** temperature values are set to `273 K` and `310 K`, respectively.

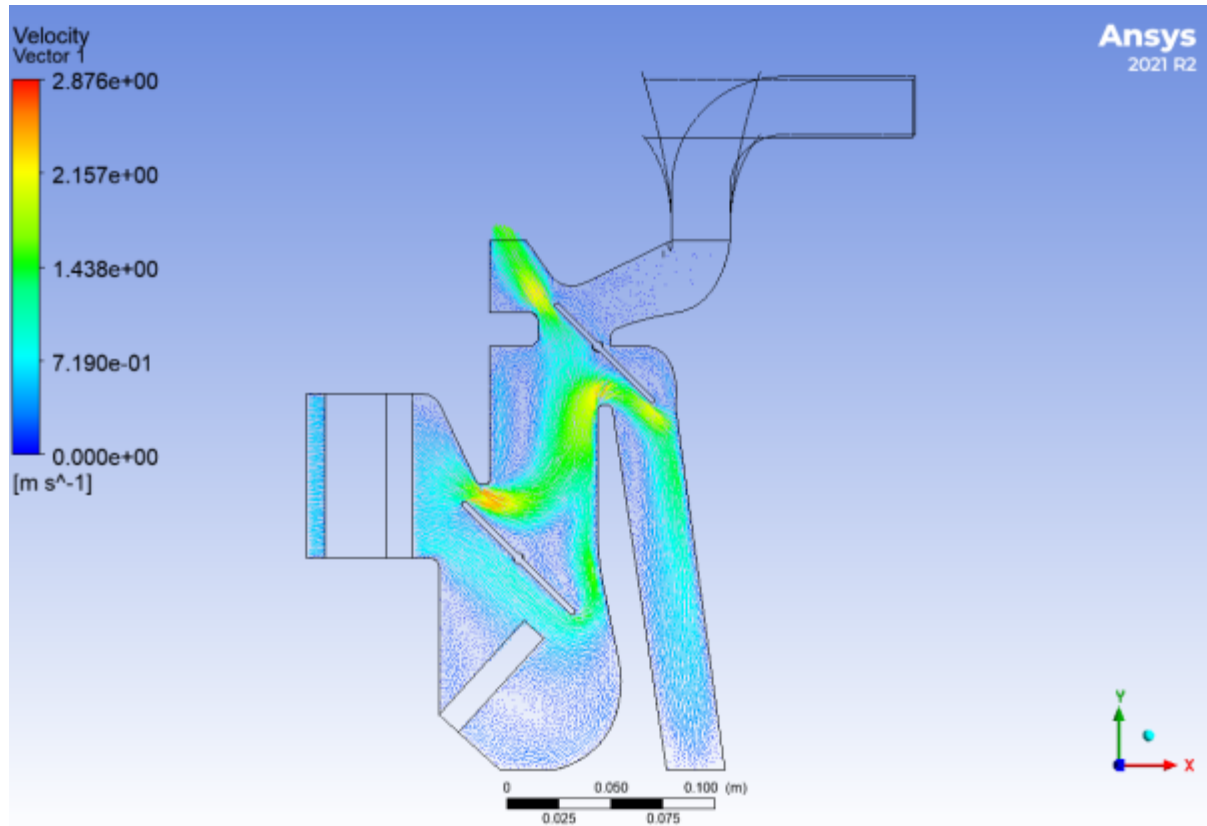**Figure 2.21: Vectors Colored by Temperature (DP1)**
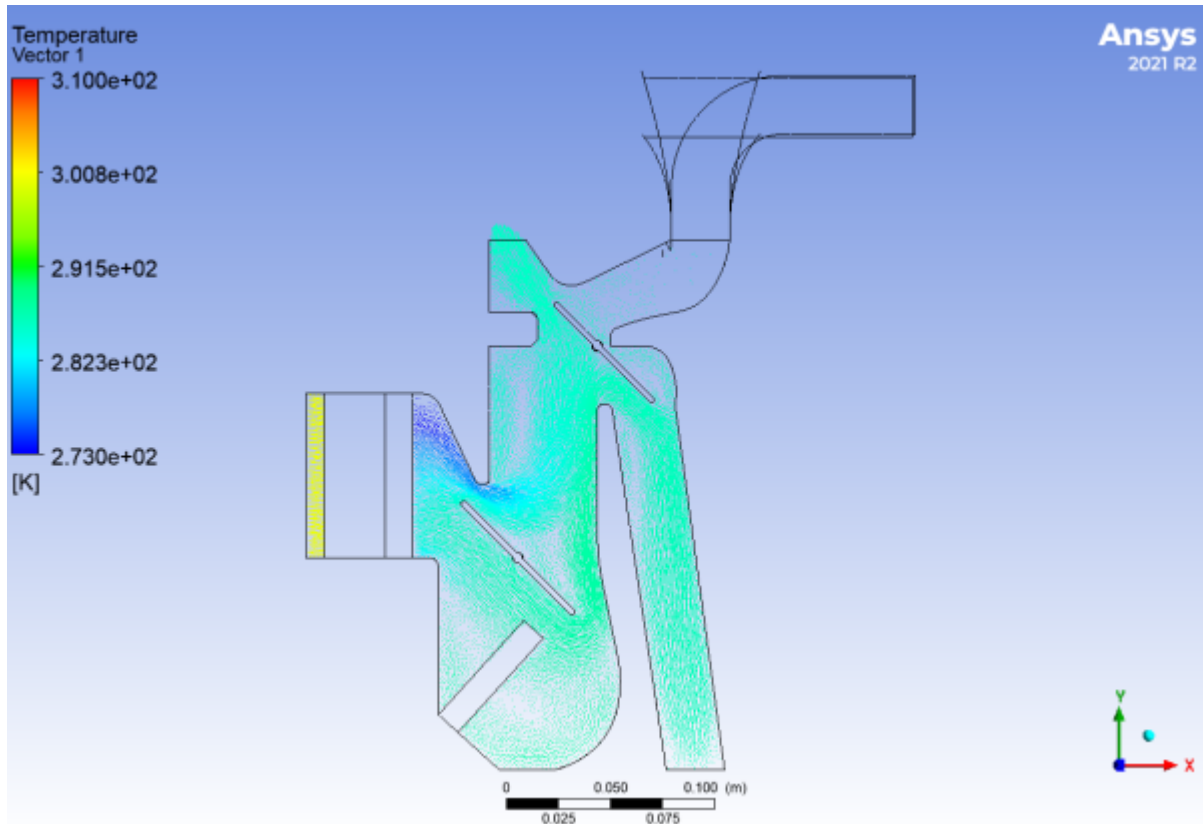
d. View the vector plot colored by pressure. Ensure that **Range** in the **Color** tab is set to **Global**.

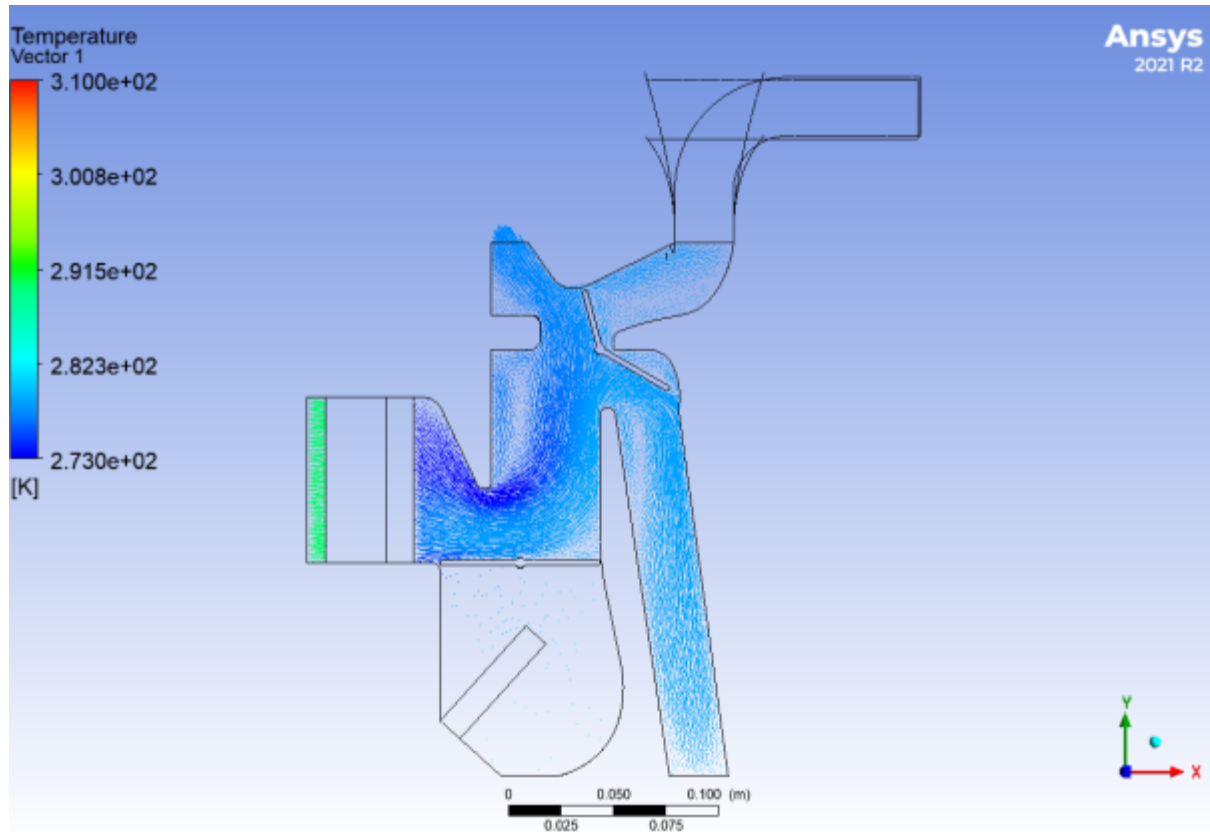**Figure 2.22: Vectors Colored by Pressure (DP1)**
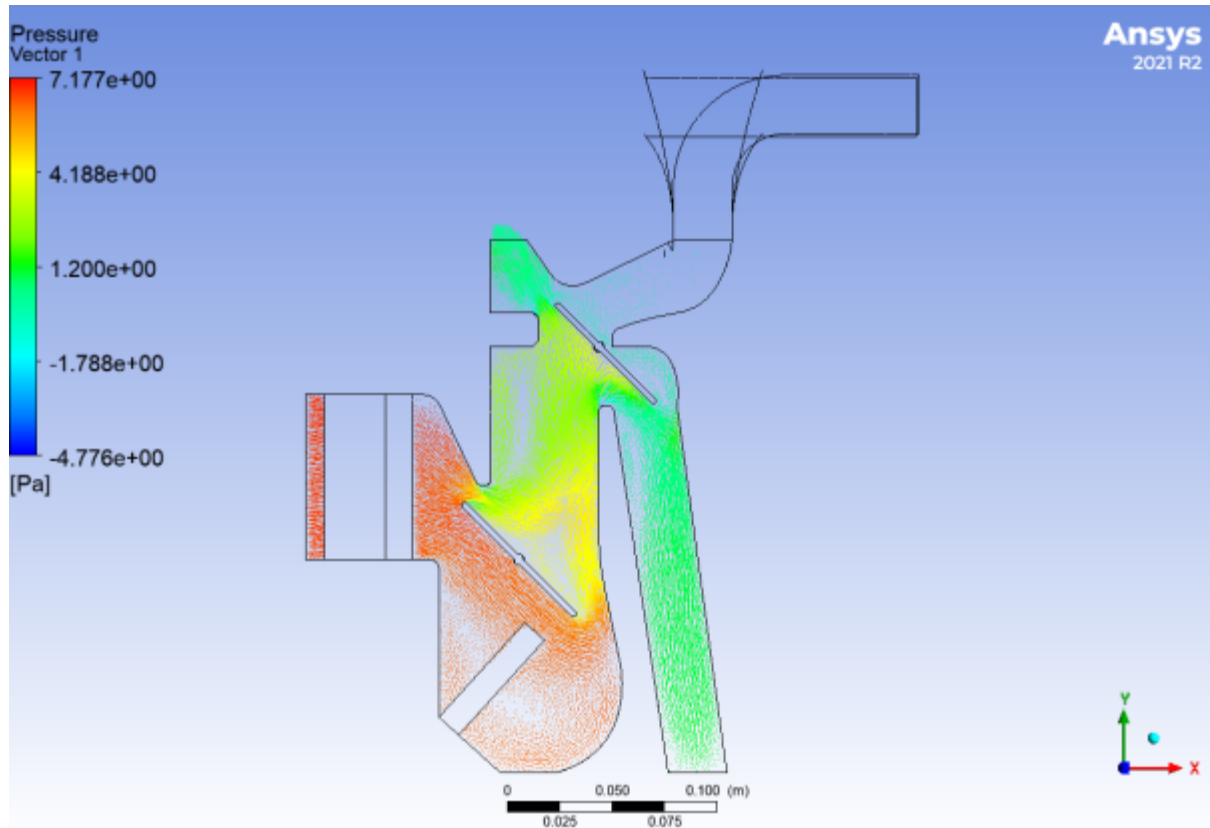
e. View the vector plot colored by velocity. Ensure that **Range** in the **Color** tab is set to **Global**.
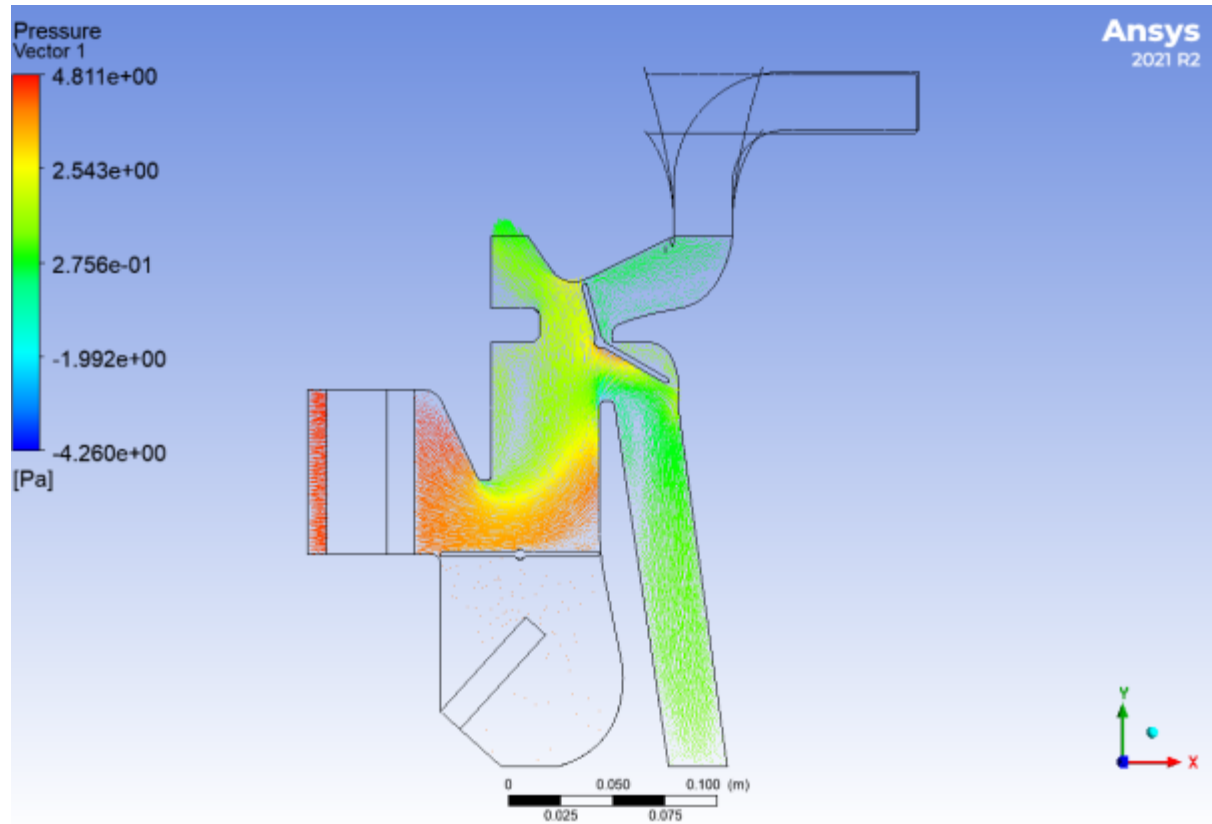
Figure 2.23: Vectors Colored by Velocity (DP1)

f.   When you are finished viewing results of the design point DP1 in Ansys CFD-Post, select **File → Close CFD-Post** to quit Ansys CFD-Post and return to the Ansys Workbench **Project Schematic**, and then select **File → Exit** to exit from Ansys Workbench.

2.   Study the results of the second design point (DP2).

a.   Open the Ansys Workbench project for the second design point (DP2).

In your current working folder, double-click the `fluent-workbench-param_dp2.wbpj` file to open Ansys Workbench.

b.   Open CFD-Post by double-clicking the **Results** cell in the Project Schematic for the Fluid Flow (Fluent) analysis system.

c.   View the vector plot colored by temperature. Ensure that **Range** in the **Color** tab is set to **User Specified** and the **Min** and **Max** temperature values are set and the **Min** and **Max** temperature values are set to `273 K` and `310 K`, respectively.
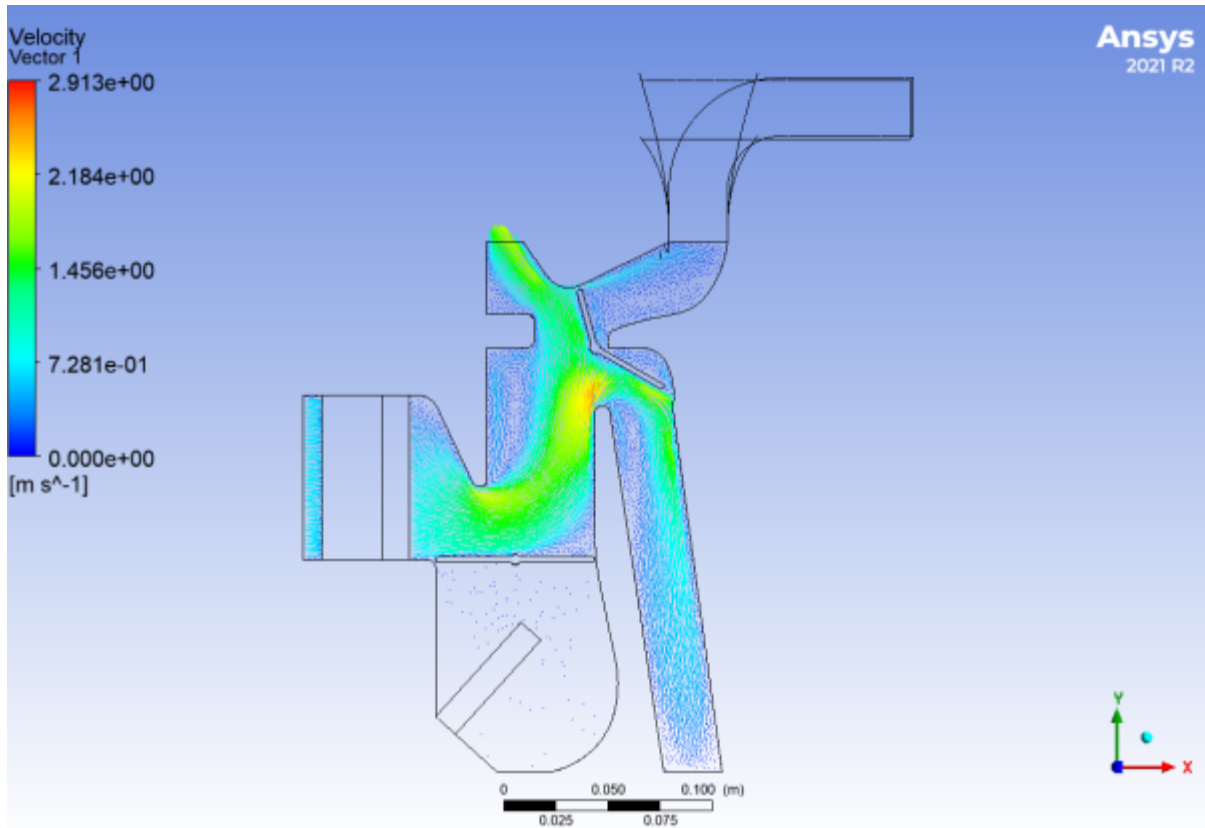
Figure 2.24: Vectors Colored by Temperature (DP2)

d. View the vector plot colored by pressure. Ensure that **Range** in the **Color** tab is set to **Global**.

Figure 2.25: Vectors Colored by Pressure (DP2)

e. View the vector plot colored by velocity. Ensure that **Range** in the **Color** tab is set to **Global**.

Figure 2.26: Vectors Colored by Velocity (DP2)

3. When you are finished viewing results in Ansys CFD-Post, select **File** → **Close CFD-Post** to quit Ansys CFD-Post and return to the Ansys Workbench **Project Schematic**, and then select **File** → **Exit** to exit from Ansys Workbench.

2.4.9. Summary

In this tutorial, input and output parameters were created within Ansys Workbench, Ansys Fluent, and Ansys CFD-Post in order to study the airflow in an automotive HVAC system. Ansys Fluent was used to calculate the fluid flow throughout the geometry using the computational mesh, and Ansys CFD-Post was used to analyze the results. Ansys Workbench was used to create additional design points based on the original settings, and the corresponding simulations were run to create separate projects where parameterized analysis could be performed to study the effects of variable angles of the inlet valves, velocities, and temperatures. Also, note that simplified solution settings were used in this tutorial to speed up the solution time. For more improved solution accuracy, you would typically use denser mesh and higher order discretization for all flow equations.

112